111-39
131798
P.56

NASA Contractor Report 189152

Coupled Structural/Thermal/Electromagnetic Analysis/Tailoring of Graded Composite Structures

Third Annual Status Report

R.L. McKnight, H. Huang, M. Hartle,
R. Holt, and D. Carpenter
General Electric Aircraft Engines
Cincinnati, Ohio

April 1992

Prepared for
Lewis Research Center
Under Contract NAS3-24538

NASA
National Aeronautics and
Space Administration

Foreword

This report has been prepared to expedite early dissemination of the information generated under the contract. The data and conclusions must be considered preliminary and subject to change as further progress is made on this program. This is a progress report covering the work done during the fourth 12 months of the contract; it is not a final report. The NASA Program Manager is Dr. C.C. Chamis.

Table of Contents

List of Illustrations

Figure **Page**

List of Tables

Nomenclature

AID	-	Automatic Improvement of Design
$\vec{B}$	-	Magnetic Flux Density
C	-	Specific Heat
[C]	-	Damping Matrix
COLSOL	-	Linear Equations Solution Routine
COLSOV	-	Version of COLSOL for Heat Transfer
CSTEM	-	Coupled Structural/Thermal/Electromagnetic Analysis/Tailoring of Graded Composite Structures
ESMOSS	-	Engine Structures Modeling Software System
FEM	-	Finite Element Method
$\{F_B\}$	-	Body Force Vector
$\{F_I\}$	-	Initial Strains Vector
$\{F_{NL}\}$	-	Nonlinear Strains Vector
$\{F_S\}$	-	Surface Tractions Vector
GEAE	-	GE Aircraft Engines
$G(x,y,z)$	-	Value of Function at any Point (x,y,z)
$G(x_i,y_i,z_i)$	-	Value of Function at Node Point i
$\vec{H}$	-	Magnetic Field Strength
h	-	Convection Coefficient
H_i	-	Isoparametric Shape Functions
$\vec{J}$	-	Magnetic Current Density
[K]	-	Stiffness Matrix
$[K_t]$	-	Magnetic Permeability Matrix
[M]	-	"Consistent" Mass Matrix

Nomenclature (Concluded)

n	-	Number of Nodes in an Element
$\{Q_B\}$	-	Element Heat Generation Vector
$\{Q_C\}$	-	Nodal Heat Flow Vector
$\{Q_S\}$	-	Boundary Heat Flow Vector
$\{q\}$	-	Element Heat Capacity Vector
STAEBL	-	Structural Tailoring of Engine Blades
TB	-	Boundary Temperature
W	-	Frequency
α	-	Surface Heat Transfer Coefficient
γ	-	Reciprocal Permeability
λ_i	-	ith Eigenvalue
μ	-	Magnetic Permeability
ρ	-	Resistivity
ϕ_i	-	ith Eigenvector

1.0 Introduction

This technical program is the work of the Life Analysis and Methods Technology Section of GE Aircraft Engines in response to NASA RFP 3-537260, "Coupled Structural/Thermal/Electromagnetic (CSTEM) Analysis/Tailoring of Graded Composite Structures." The overall objective of this program is to develop and verify analysis and tailoring capability for graded composite engine structures taking into account the coupling constraints imposed by mechanical, thermal, acoustic, and electromagnetic loadings.

The first problem that was attacked is the development of finite elements capable of accurately simulating the structural/thermal/electromagnetic response of graded composite engine structures. Because of the wide diversity of engine structures and the magnitudes of the imposed loadings, the analysis of these is very difficult and demanding when they are composed of isotropic, homogeneous materials. The added complexity of directional properties which can vary significantly through the thickness of the structures will challenge the state of the art in finite element analysis. We are applying AE's 25 years of experience in developing and using structural analysis codes and the exceptional expertise of our University consultants toward the successful conclusion of this problem. To assist in this, we drew heavily on previously funded NASA programs.

We built on NASA programs NAS3-23698, 3D Inelastic Analysis Methods for Hot Section Components, and NAS3-23687, Component Specific Modeling, in the development of the plate and shell elements. In addition to these two programs, we drew on NAS3-22767, Engine Structures Modeling Software System (ESMOSS), and NAS3-23272, Burner Liner Thermal/Structural Load Modeling, in Task III when we generated a total CSTEM Analysis System around these finite elements. This guarantees that we are using the latest computer software technology and produced an economical, flexible, easy to use system.

In our development of a CSTEM tailoring system, we built on NASA Program NAS4-22525, Structural Tailoring of Engine Blades (STAEBL) and AE program, Automatic Improvement of Design (AID), in addition to the program system philosophy of ESMOSS. Because of the large number of significant parameters and design constraints, this tailoring system will be invaluable in promoting the use of graded composite structures.

All during this program, we availed ourselves of the experience and advice of our Low Observables Technology Group. This will be particularly true in the Task V proof-of-concept. Their input will be used to assure the relevance of the total program.

Figure 1 shows our program and major contributions in flowchart form. This gives a visual presentation to the synergism that will exist between this program and other activities.

Figure 2 depicts an integrated analysis of composite structures currently under development in the composite users' community. The severe limitations of such a system are not highlighted because three major steps in the process are not shown. Figure 3 adds these steps. The analysis system really begins with a definition of geometry. A user then defines a

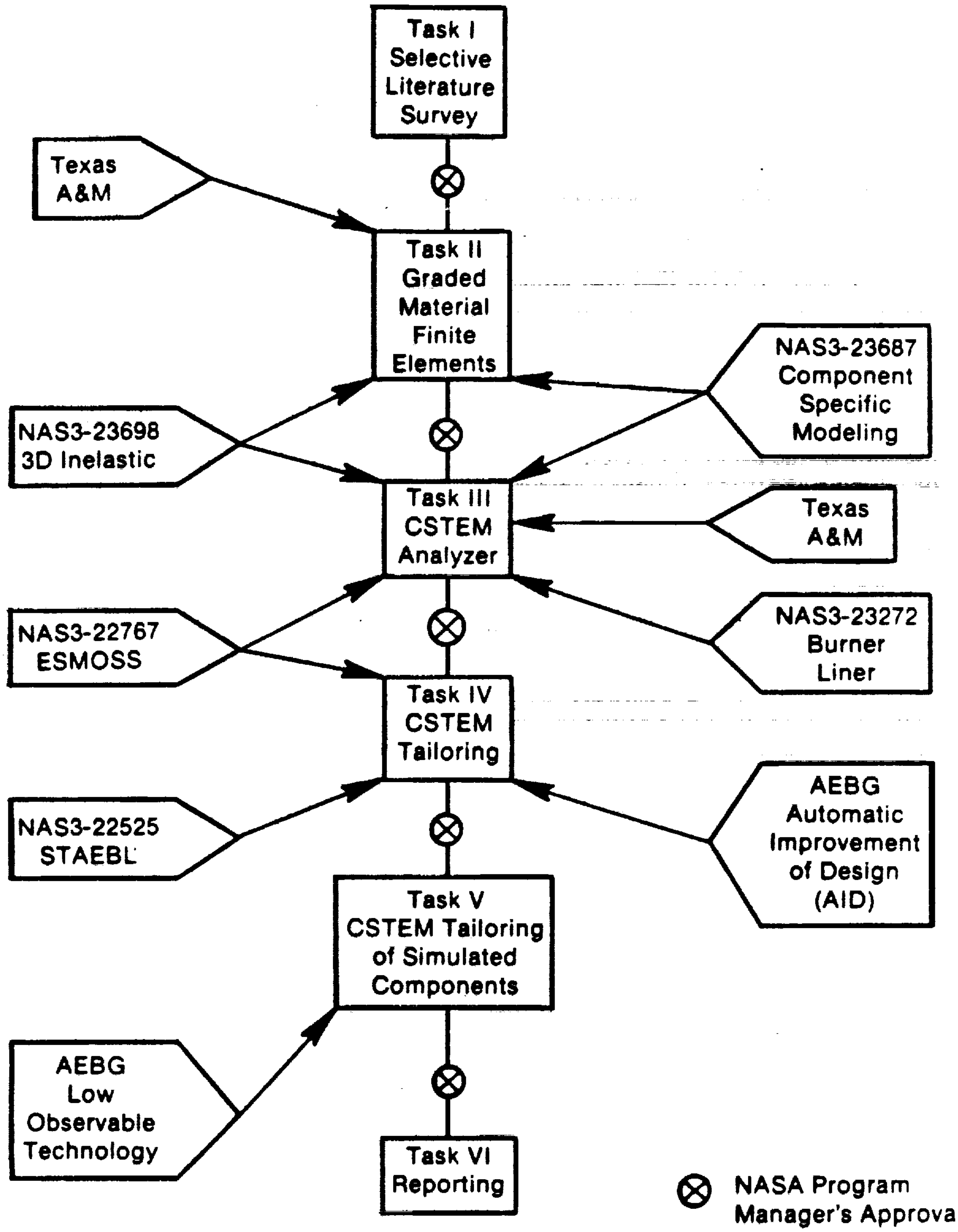


Figure 1. Program Flowchart.

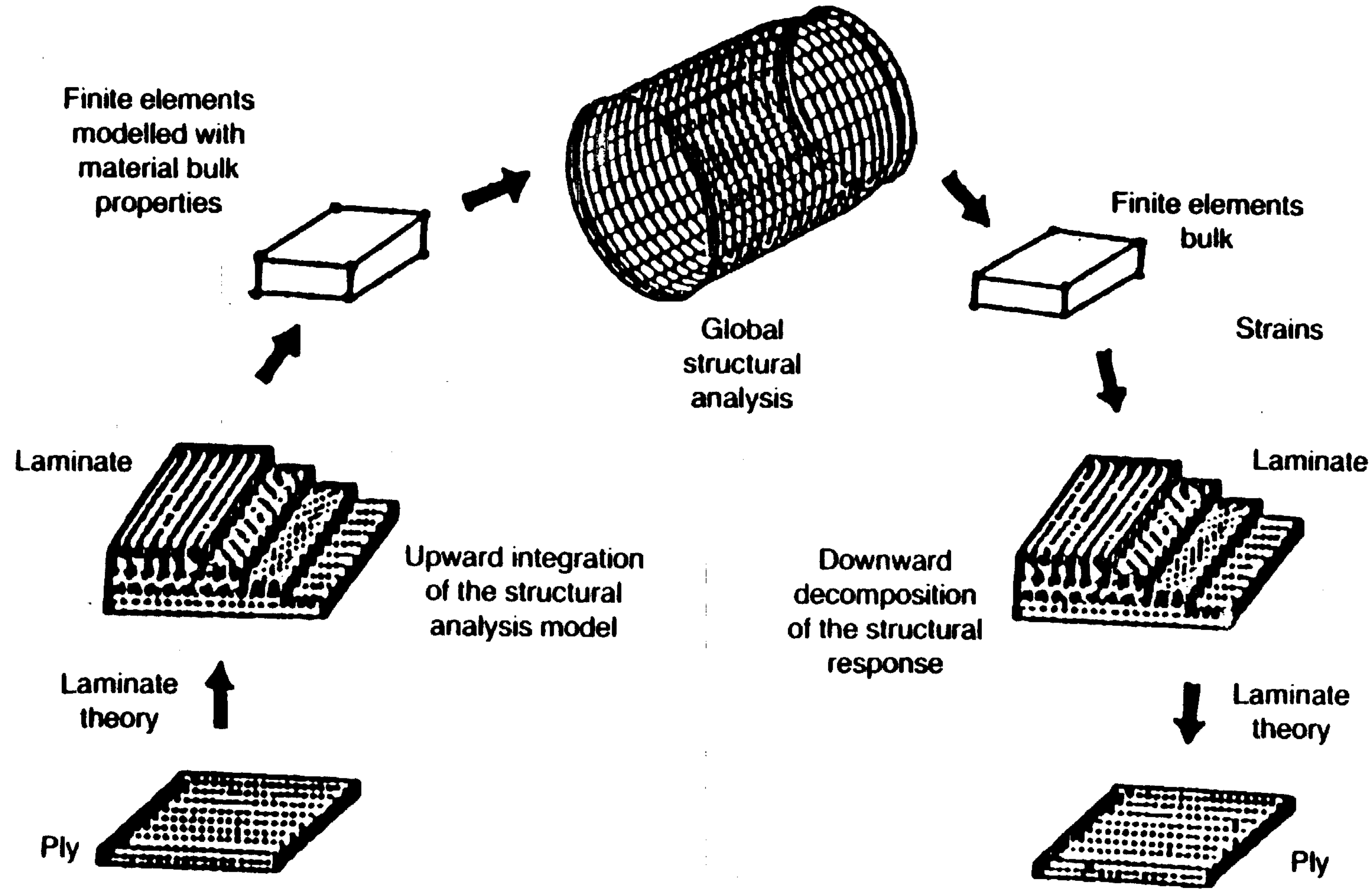


Figure 2. Composite Analysis System.

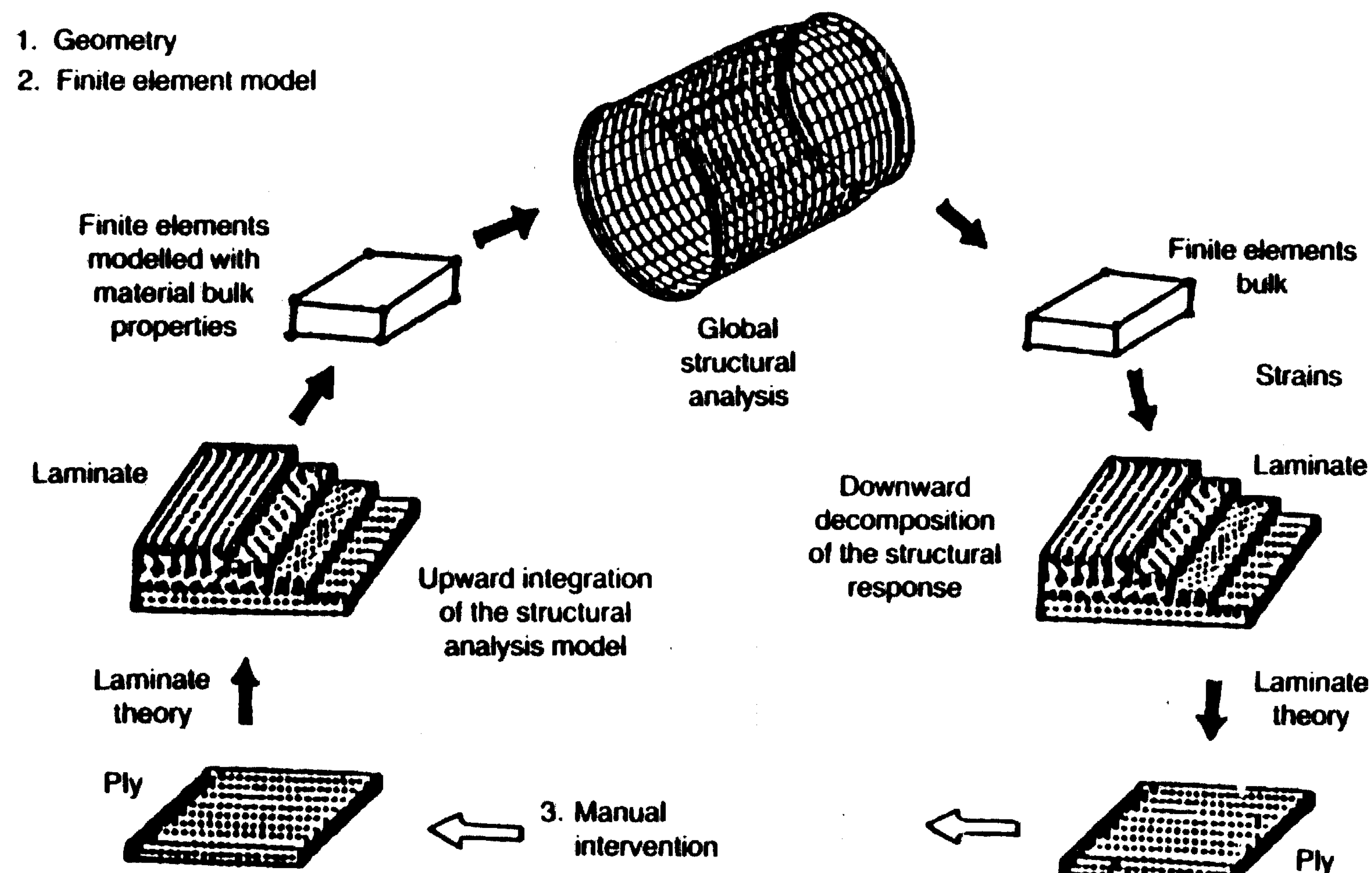


Figure 3. Total Composite Analysis System.

finite element model simulating this geometry and the anticipated loading. The process then moves to defined Step 3. One cycle through the process ends with the prediction of individual ply average stresses and strains. Now comes a significant productivity drain, namely, manual intervention to evaluate these stresses and strains against strength and durability limits. Based on this, the user must decide to (1) change the finite element model, (2) change the composite laminate, (3) both of the above, or (4) stop here.

Obviously, there is a considerable cost savings to be obtained by selecting Number 4. The CSTEM system will obviate the reasons for selecting Number 4. This system, shown in Figure 4, begins with the definition of geometry, as before, but then proceeds to a definition of master regions which contain all of the necessary information about geometry, loading, and material properties. Step 3 is a constitutive model which develops the necessary structural, thermal, and electromagnetic properties based on a micromechanics approach. Furthermore, this constitutive model contains the logic to generate the global finite element model based on the variation of the properties, as depicted in Figure 5. Using a nonlinear incremental technique, those global models are solved for their structural, thermal, and electromagnetic response. Based on this response the global characteristics are evaluated, with convergence criteria and decisions made on remodeling. Once the global characteristics meet the accuracy requirements, the local characteristics are interrogated and decisions made on remodeling because of strength, durability, or hereditary effects. Once this cycle has been stabilized, optimization is performed based on design constraint.

1.1 Executive Summary

"CSTEM" is the acronym for the computer program being developed under the NASA contract, "Coupled Structural/Thermal/Electromagnetic Analysis/Tailoring of Graded composite Structures." The technical objectives for this program are to produce radar signal transparent structures having high structural performance and low cost. The multidisciplines involved are all highly nonlinear. They include anisotropic, large deformation structural analysis, anisotropic thermal analysis, anisotropic electromagnetic analysis, acoustics, and coupled discipline tailoring. The CSTEM system is a computerized multidiscipline simulation specialized to the design problems of radar absorbing structures. The enabling technical capabilities are implemented in a special 3D finite element formulated to simultaneously tailor the geometrical, material, loading and environment complexities of radar transparent structures for cost effective optimum performance.

In each enabling technical discipline a decoupled stand-alone 3D finite element code has been developed. An executive program with controlling iterative solution techniques performs the nonlinear coupling among the participating technical disciplines. A geometry and finite element model generator specialized for graded composites has been developed as an intimate part of this analysis system.

The structural analyzer is built around the 8-, 16-, and 20-noded isoparametric finite elements with emphasis on the 20 noded. A graded composite constitutive model has been developed for these elements which uses the NASA Lewis program ICAN as a subroutine to

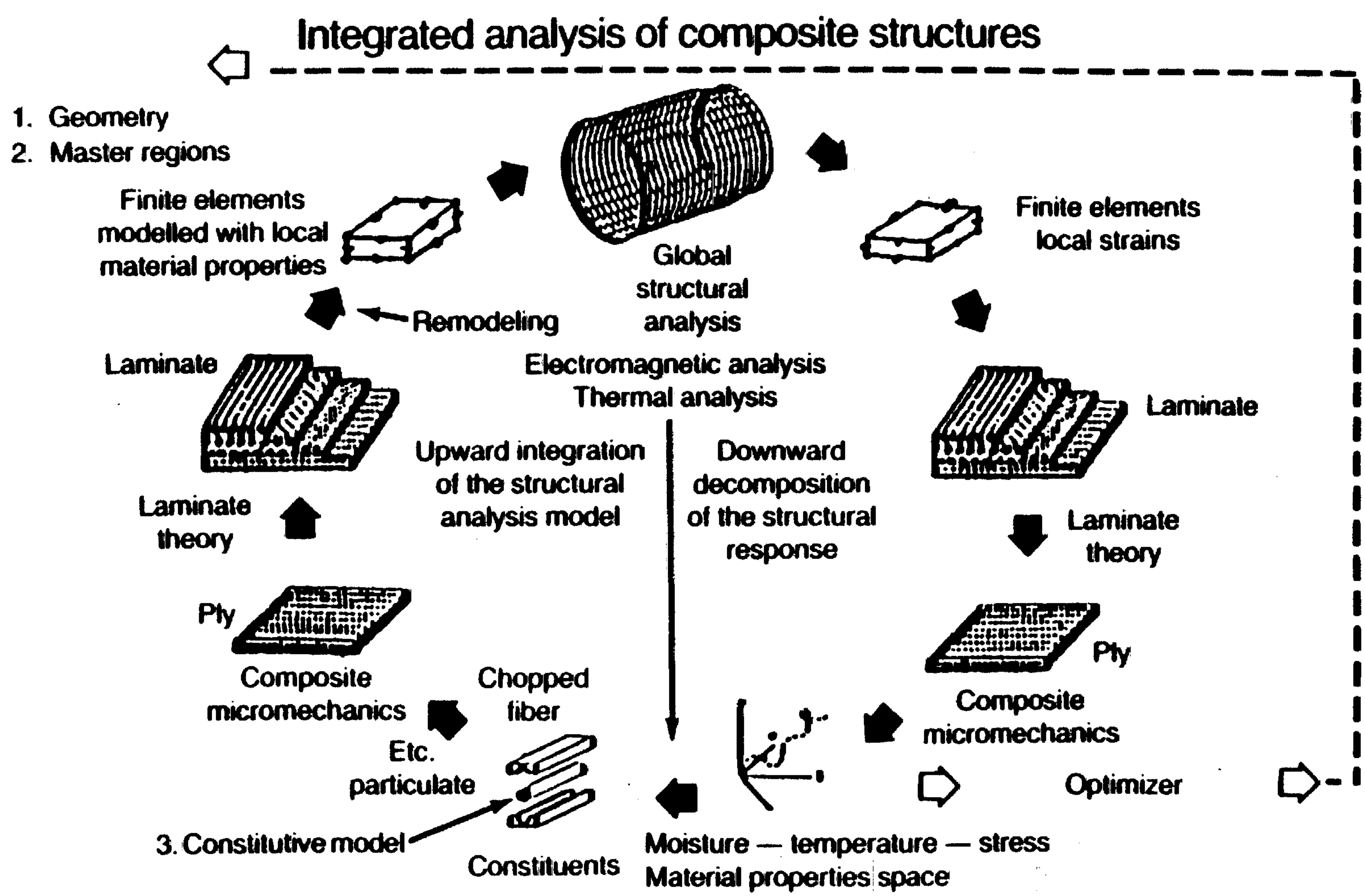


Figure 4. CSTEM System.

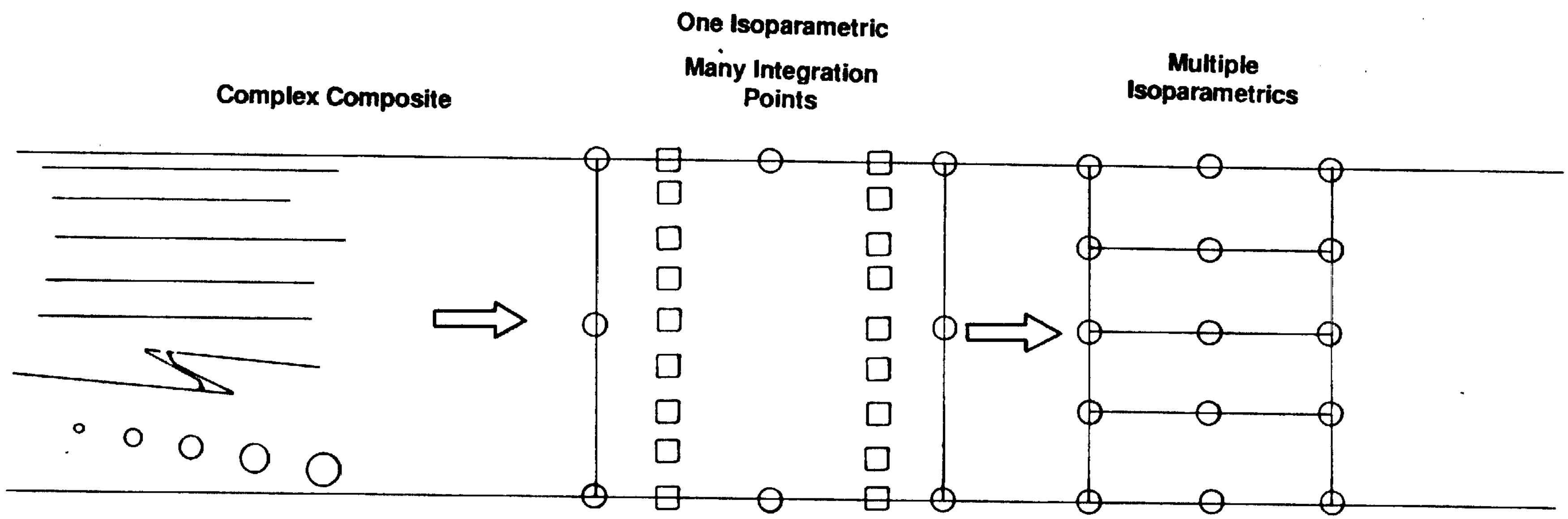


Figure 5. Constitutive Model - Structural Model Interaction.

perform composite micromechanics and supply, to the analyzer, the requisite composite properties. The composite stiffness gradient controls the finite element definition of a structure with two major parameters to vary the number of elements through the thickness and the number of numerical quadrature points within an element. A unique set of local stiffness characteristics is developed for each numerical integration point. Integration of these local characteristics over the volume of the element provides total element simulation of composite structures including such effects as twist-bend coupling.

The structural analyzer also performs large deformation analysis using a unique incremental updated Lagrangian approach with iterative refinement. Testing of this capability against classical large deformation problems has shown it to be both more accurate and more economical than available alternatives. Connected with this technical capability is a deformed position eigen-analyses capability. All or selected portions of the nonlinear stiffness terms can be incorporated into these eigenanalyses. This capability has been checked out against available test data and other computer codes.

These capabilities have been combined with an optimizer to perform a totally automated integrated analysis of graded composite structures. A final task in the program will be a design demonstration of the tailoring capabilities of the CSTEM system.

In order to reach the thermal analysis goals set for CSTEM, the same 8-, 16-, and 20-noded isoparametric finite elements are utilized. Four heat transfer solution options are available: linear steady state, nonlinear steady state, linear transient, and nonlineaar transient. To overcome the previous economic penalty associated with finite element vis-a'-vis finite difference heat transfer, a unique solution technique is employed. This is a Newton-Raphson iterative technique with right hand side pseudo-fluxes. The code will perform the heat transfer analysis of a thermally anisotropic material considering conduction, convection, and radiation. Table 1 lists the parameters involved.

Routines for the calculation of absorption of electromagnetic waves have been written and checked out. The first method developed was based on a data bank of absorptivity values for given material types specified at discrete values of temperatures, frequency, and polarization angle. The information for a specific material or materials is read from the data bank file into arrays which are used by the program. The absorptivity is linearly interpolated from these discrete values to the local values of temperature, frequency and polarization angle.

Calculations are made for one given frequency, orientation, and path of an electromagnetic wave at a time. Multiple frequencies and/or different paths travelled through the structure are handled by separate calculations for the different parameters.

A wave coordinate system is associated with the electromagnetic wave. This wave coordinate system is defined such that the direction of propagation is along the positive y axis and polarization is measured from the positive x axis.

Table 1. Thermal Analyzer.

Thermal Parameters and Boundary Conditions	Steady State		Transient	
	Linear	Nonlinear	Linear	Nonlinear
• Temperature	T	T	T(t)	T(t)
• Time	---	---	t	t
• Thermal Conductivity	kij	kij(T)	kij(t)	kij(T,t)
• Convection Coefficient	h	h(T)	h(t)	h(T,t)
• Internal Heat Generation	Q_i	Q_i	Q_i (t)	Q_i (t)
• Surface Heat Flux	Q_s	Q_s	Q_s (t)	Q_s (t)
• Convection Boundary	Q_c	Q_c	Q_c (t)	Q_c (t)
• Specified Nodal Temperatures	T_s	T_s	T_s (t)	T_s (t)
• Heat Capacity	---	C_p (T)	---	C_p (T,t)
• Radiation Emissivity	---	ε (T)	---	ε (T,t)
• Viewing Factor	---	f	---	f (t)

The element face upon which the wave impinges is input and the angle of incidence of the wave is calculated from the dot product of the inward normal of the layer subsurfaces and the positive y axis of the wave coordinate system. The angle of incidence must be between 0 and 90 degrees in absolute value. The temperature is calculated using the element shape functions.

The polarization angle is calculated using the dot product between the projection of the wave polarization on the layer midsurface and the material principle direction. The polarization angle is calculated as being between 0 and 90 degrees.

As a second alternative, a method was developed that calculates the electromagnetic response and determines the reflected amounts of electromagnetic energy as well as the attenuation of the transmitted amount. This technique is based on Maxwell's equations, Snell's law, and the Fresnel formula.

A third alternative has been provided by installing the computer program WAVES as a subroutine in CSTEM. This program calculates the reflection and transmission of electromagnetic waves given a stackup sequence of materials and their electromagnetic properties.

The capability to determine acoustic characteristics due to structural vibration has been incorporated into CSTEM. The approach used determines the radiation efficiencies of a structure for each vibration mode as a function of frequency. An eigenanalysis produces the fundamental modes and mode shapes. Once the radiation efficiencies for each mode are calculated, the total sound power is obtained by a modal summation of the contribution from each mode.

CSTEM tailoring capability has been built on the STAEBL (Structural Tailoring of Engine Blades) computer program obtained from NASA Lewis. This program consists of two major modules; CONMIN, which performs optimization and ANALIZ which supplies the parameter to be optimized. The CONMIN module was abstracted from STAEBL and coupled with the CSTEM structural, thermal, electromagnetic and acoustic application modules. This was successfully debugged, verified, and validated.

Further improvements have been made to the CSTEM micromechanics routines and a unique modeling and solution technique for ply drop-off has been developed.

2.0 Technical Progress

2.1 Demonstration of Coupled Solution Capability

A simulated composite duct was generated to use in testing the coupled structural/thermal/electromagnetic solution capabilities of CSTEM. The CSTEM model generator was used to automatically develop the finite element model shown in Figure 6. Thirty-two 20-noded graded composite finite elements were generated with four composite layers (+60°, -60° +60°, -60°) in each element. The thermal problem consisted of convection at the inner surface from a temperature varying fluid and the outer wall being held at a constant temperature. Figure 7 plots the time variation of the convection coefficient and Figure 8 that of the internal environment temperature. The necessary thermal properties of the composites came from the internal ICAN data bank. The structural loading, in addition to temperature, consisted of internal pressure and an imposed duct end displacement. Figure 9 is a time history of the end displacement and Figure 10 is a time history of the internal pressure. For the electromagnetic absorption problem, simulated temperature, frequency, and polarization dependent absorption tables were supplied to the data bank (Figures 11 and 12). Figure 13 is a plot of the original and the loaded, deformed geometry. Figure 14 is a plot of the variation of the temperature through the duct thickness. Figure 15 plots the effective stress for a line of elements covering the distance from duct tip to duct root and showing the composite layer effect. Figures 16 and 17 show the variation around the circumference of the composite layer effective stresses and strain for the tip elements. Figures 18 and 19 show the same information for the root elements. The effect of the ± 60° layers is evident. This run required approximately 89 CPU seconds on the CRAY-XMP.

2.2 Alternate CSTEM Electromagnetic Technology

A different method for calculation of electromagnetic response has been included in CSTEM that determines reflected amounts of electromagnetic energy as well as calculating the attenuation of the transmitted amount. Figure 20 shows an electromagnetic wave propagating in the +Z direction of the coordinate system attached to the wave. The electric field vector points in the +X direction and the magnetic field vector points in the +Y direction for a polarization of zero degrees.

The wave equations which apply to these vectors can be obtained using Maxwell's equations, and can be written as:

$$\nabla^2 H = \gamma^2 H \text{ and } \nabla^2 E = \gamma^2 E$$

where the propagation constant, γ, can be written as:

$$\gamma = \alpha + j\beta$$

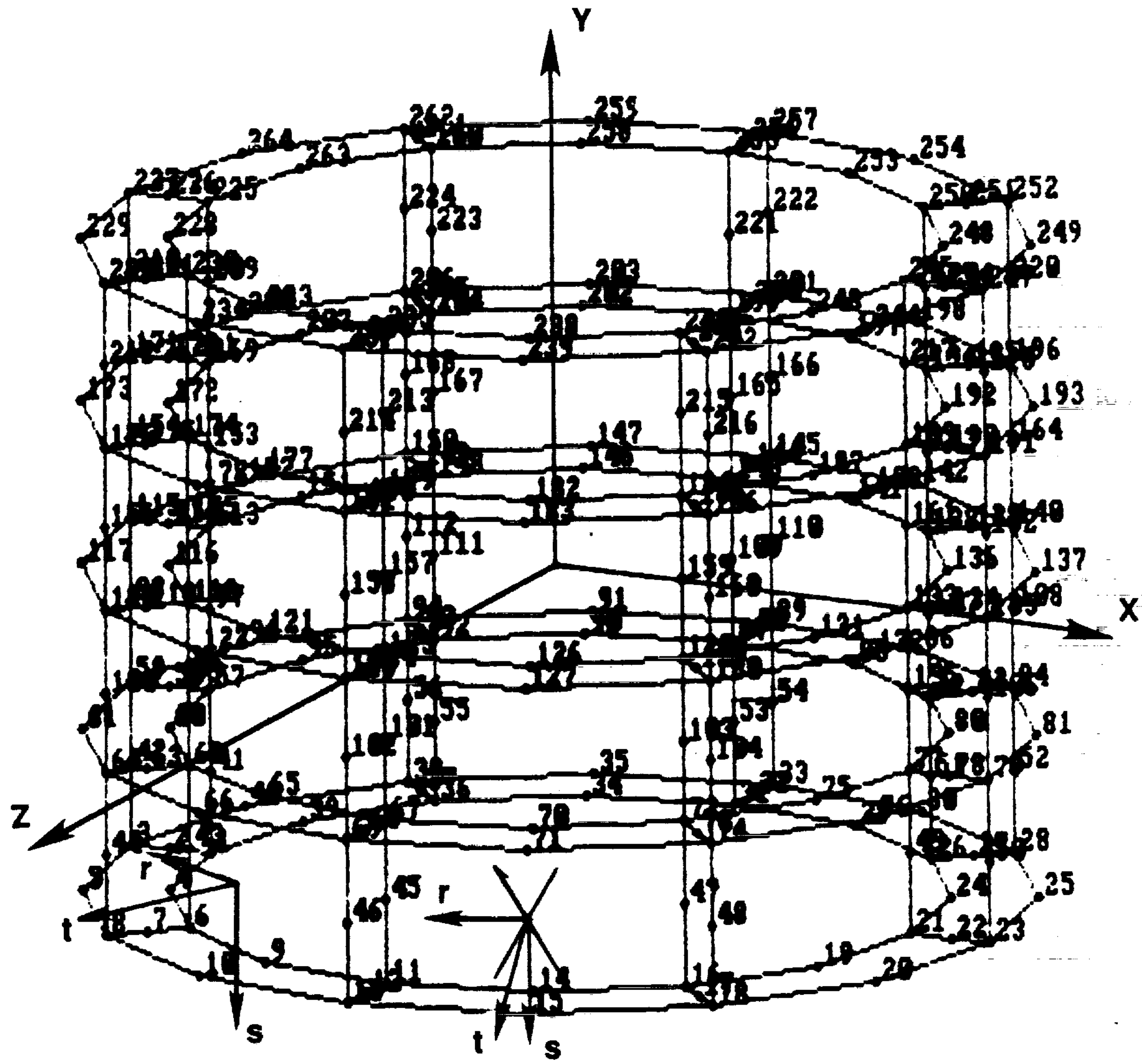


Figure 6. Composite Duct Model.

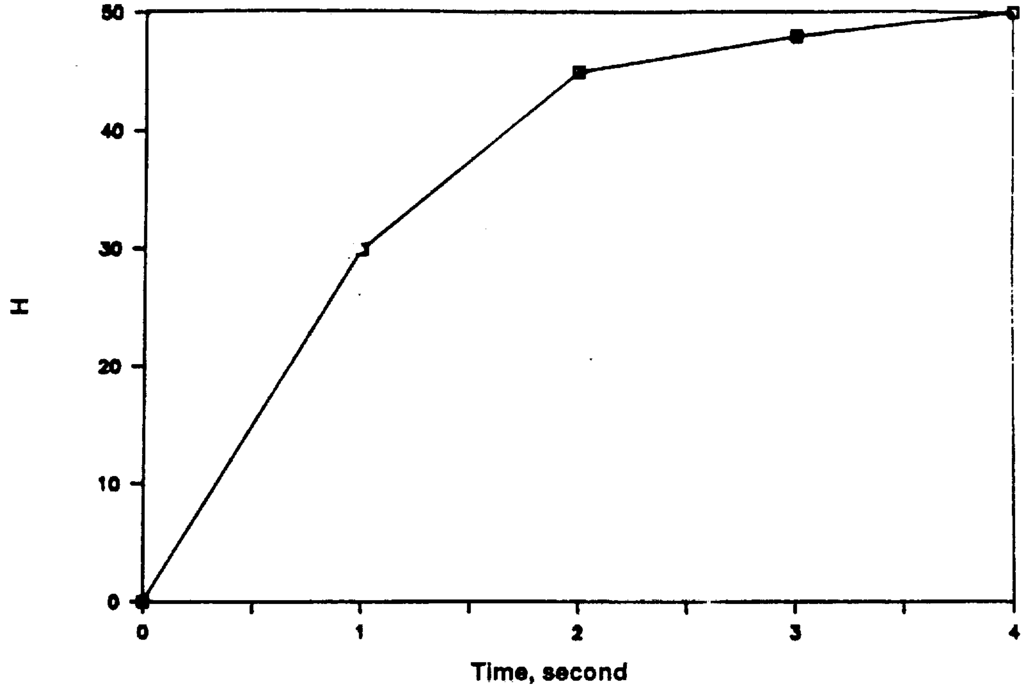


Figure 7. Convection Coefficient.

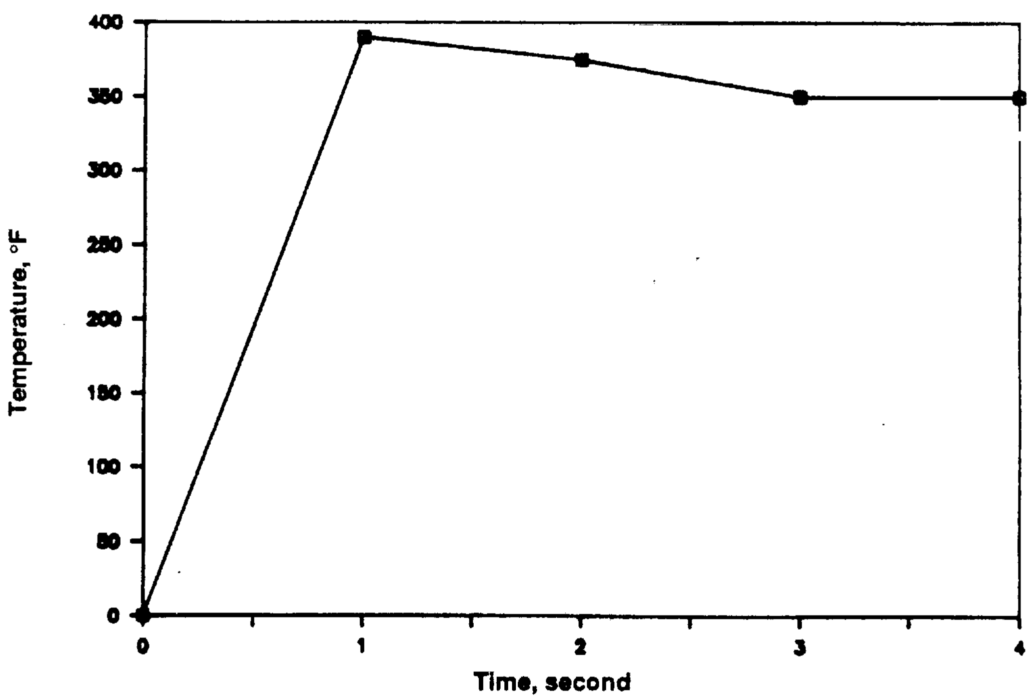


Figure 8. Environment Temperature.

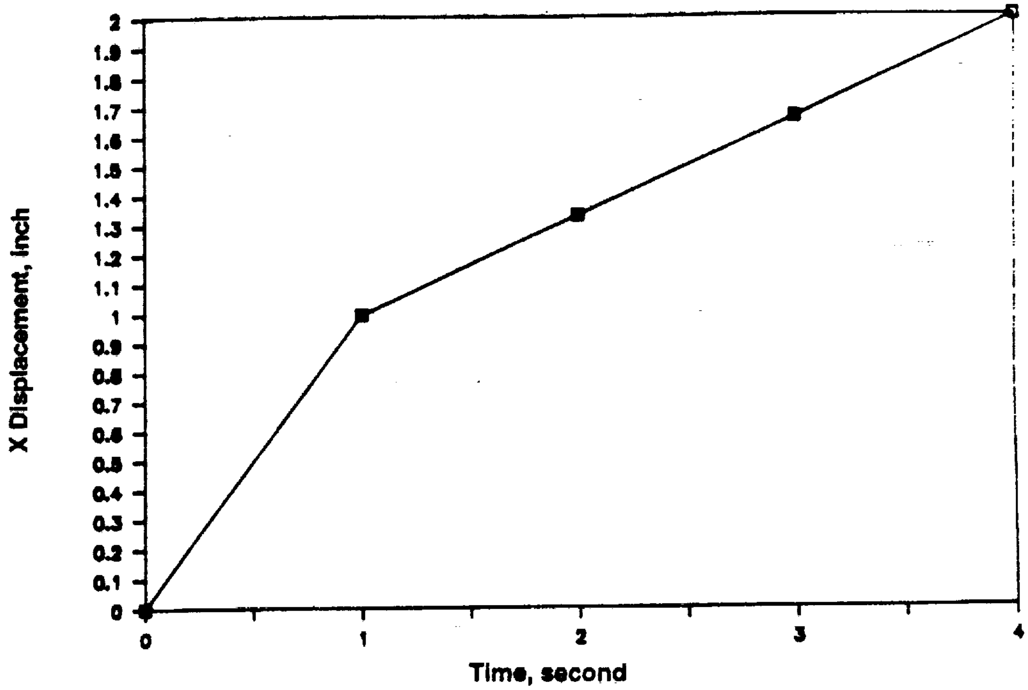


Figure 9. End Displacement.

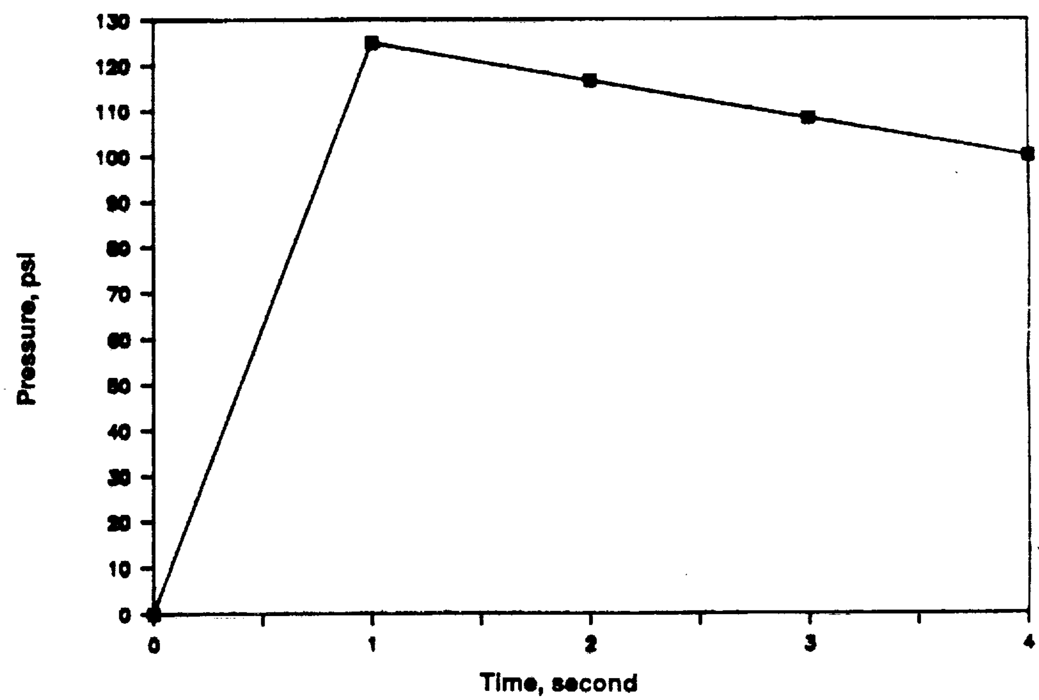


Figure 10. Internal Pressure.

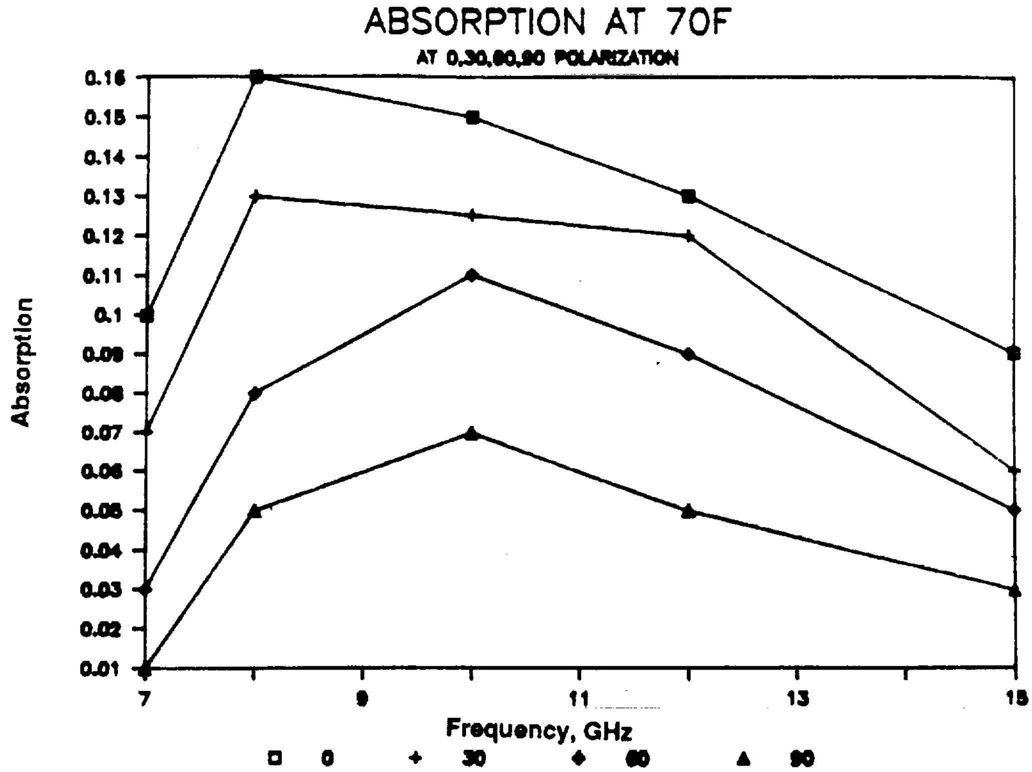


Figure 11. Absorption at 70°F.

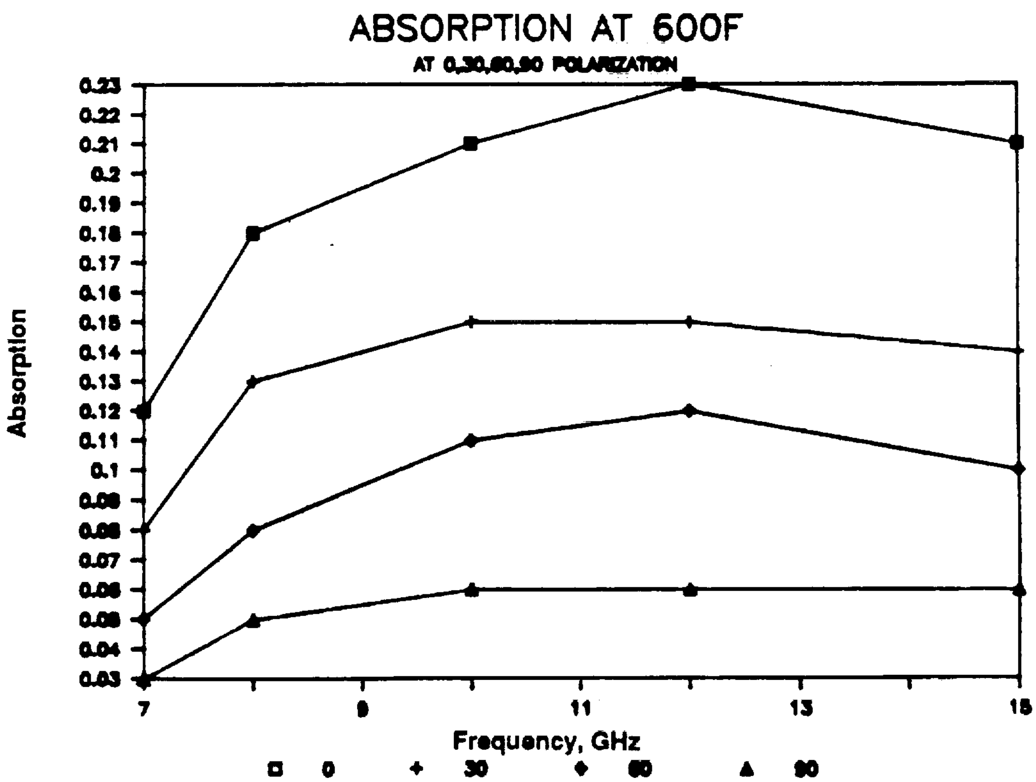


Figure 12. Absorption at 600°F.

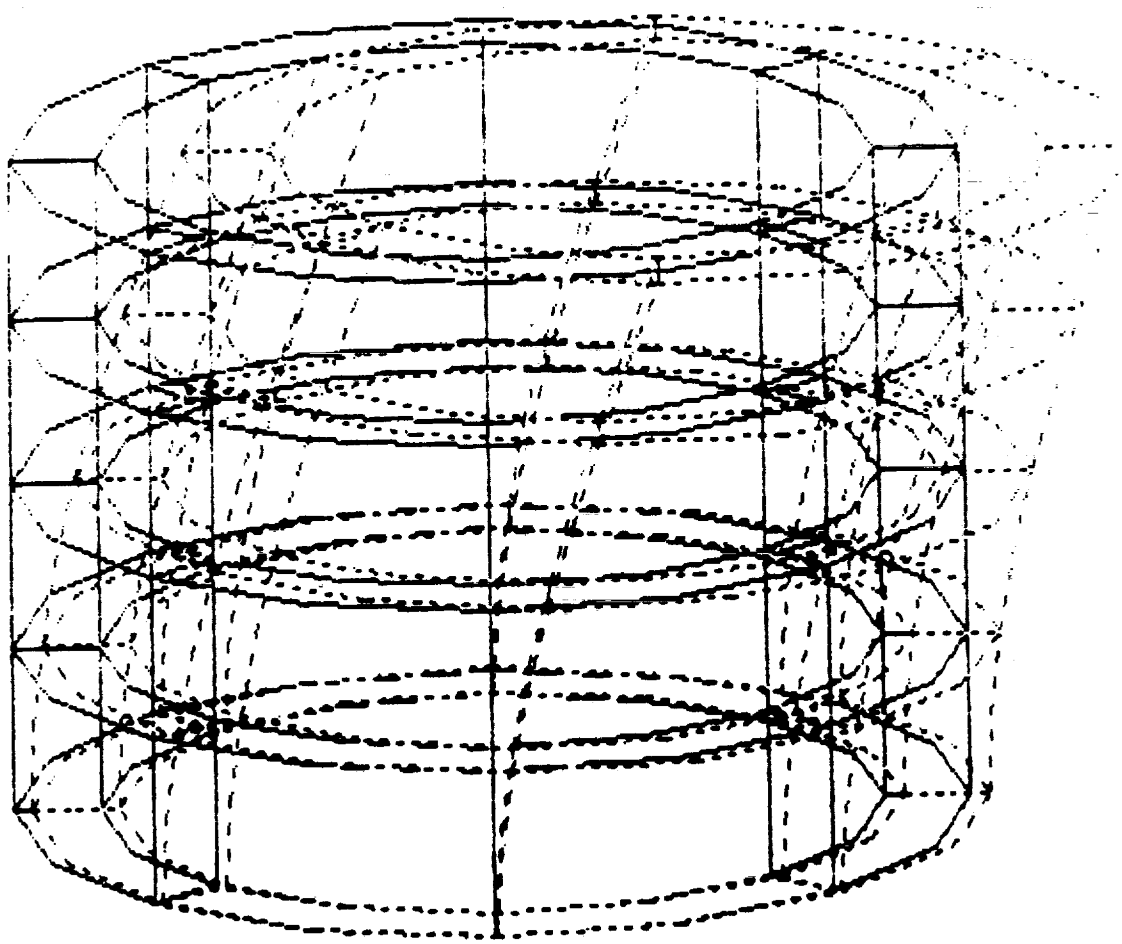


Figure 13. Original and Loaded, Deformed Geometry.

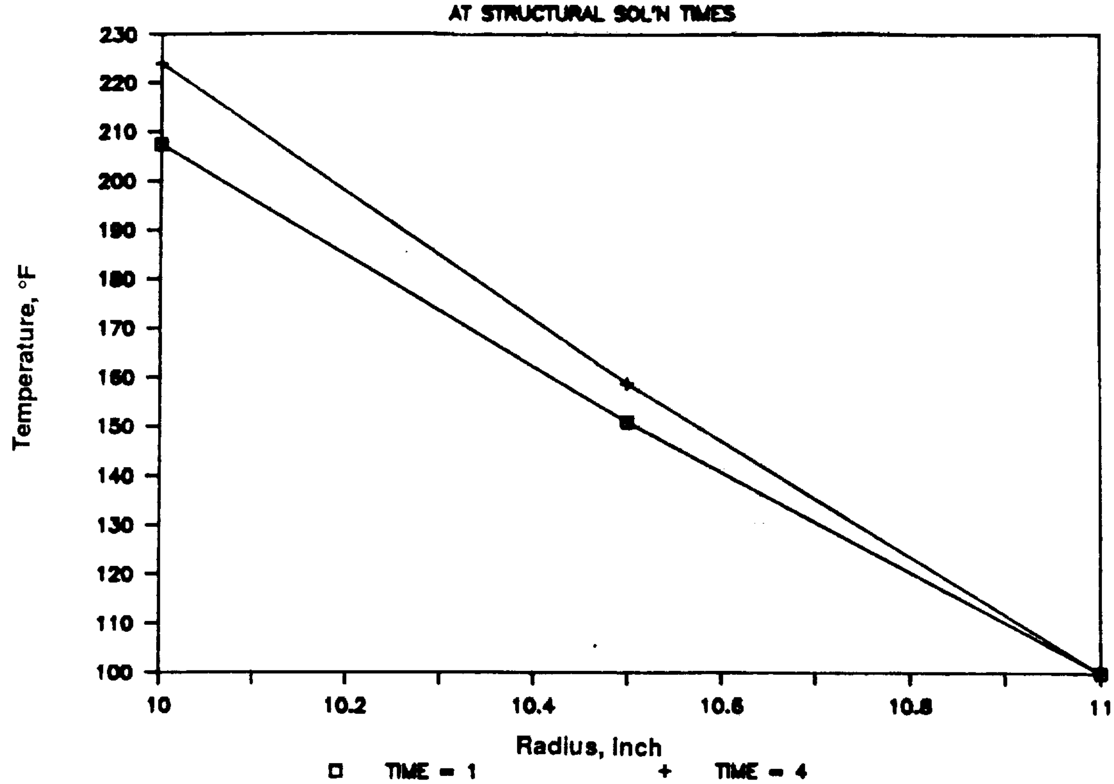


Figure 14. Temperature Thru Thickness.

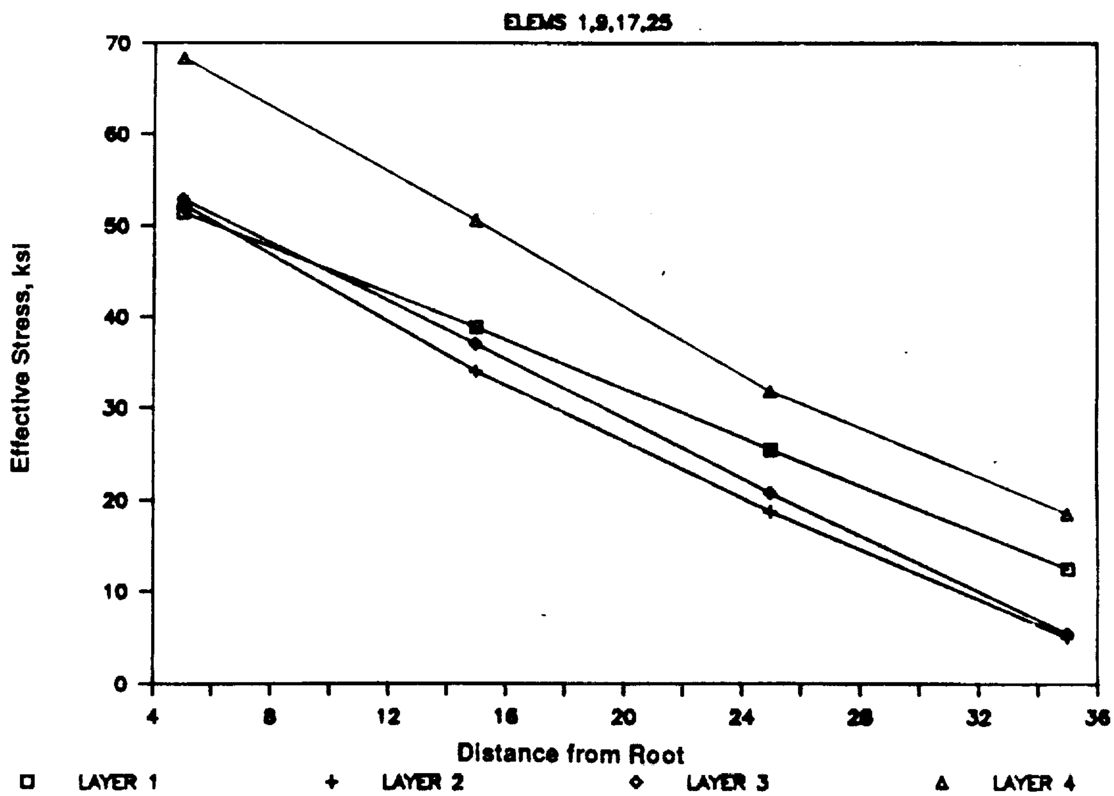


Figure 15. Effective Stress Versus Distance from Duct Root.

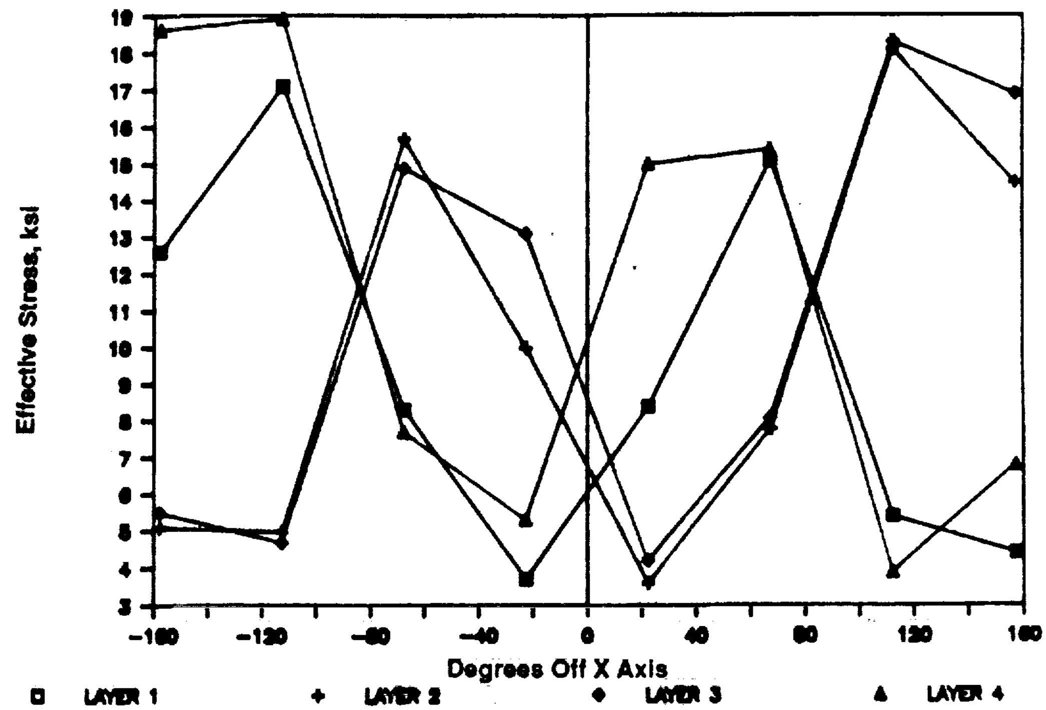


Figure 16. Effective Stress at Tip Element Centroids.

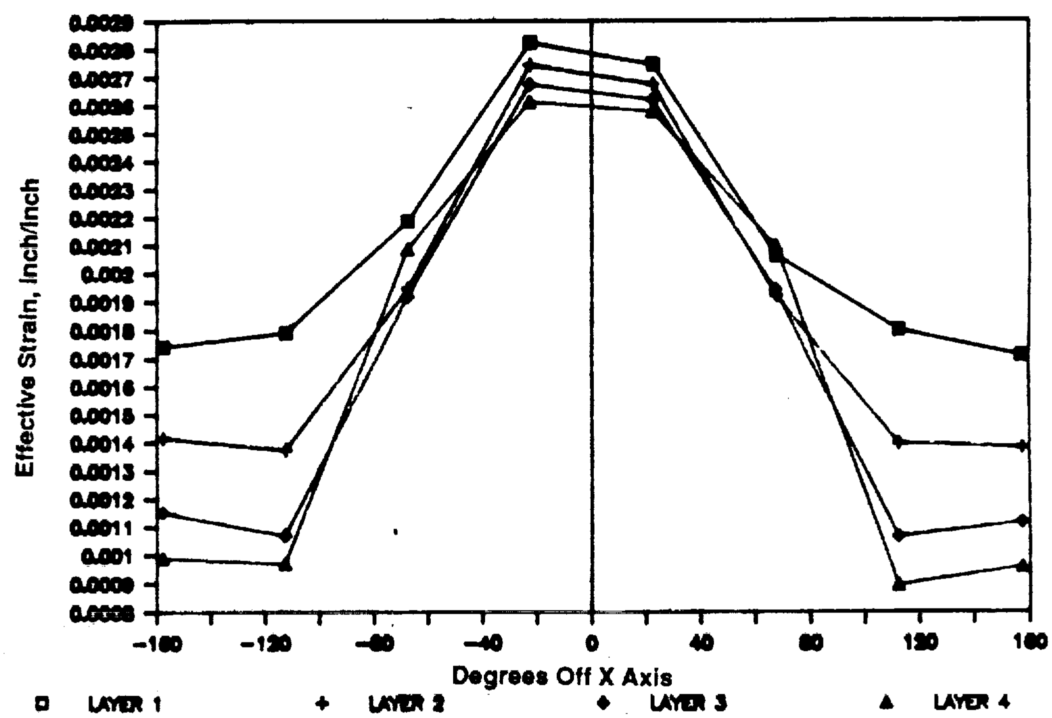


Figure 17. Effective Strain at Tip Element Centroids.

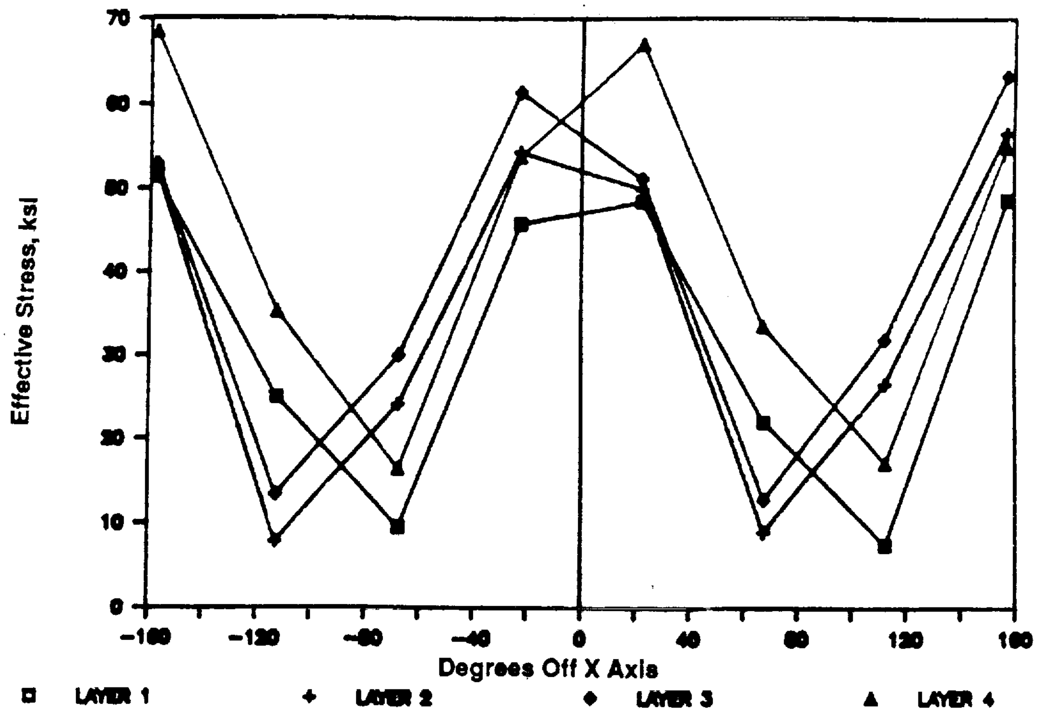


Figure 18. Effective Stress at Root Element Centroids.

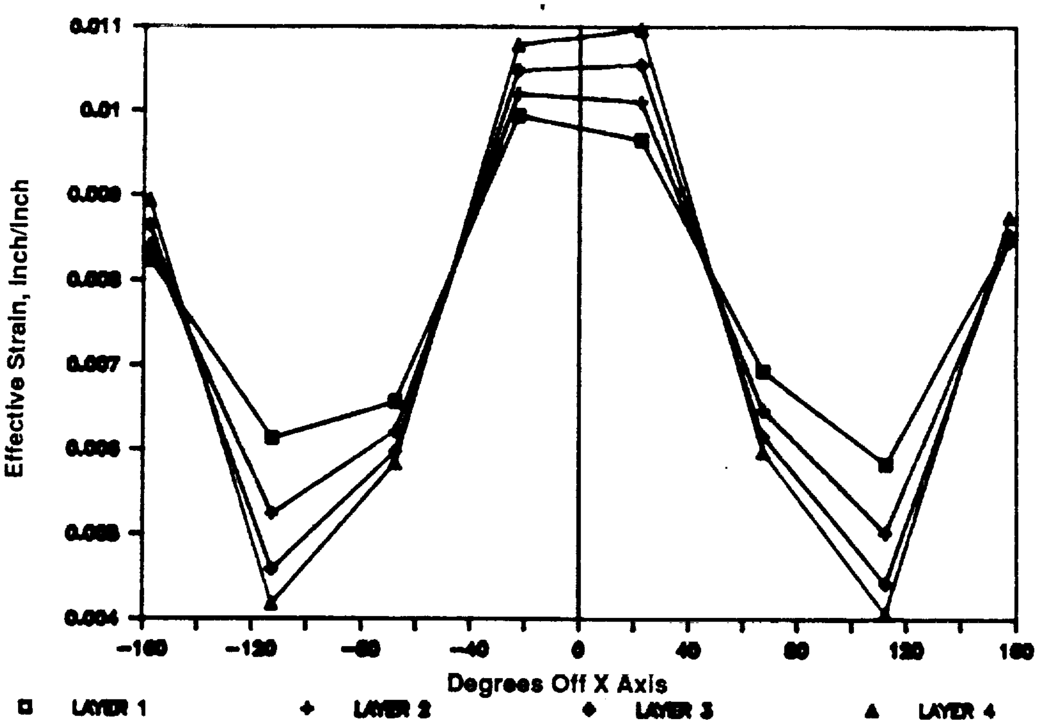


Figure 19. Effective Strain at Root Element Centroids.

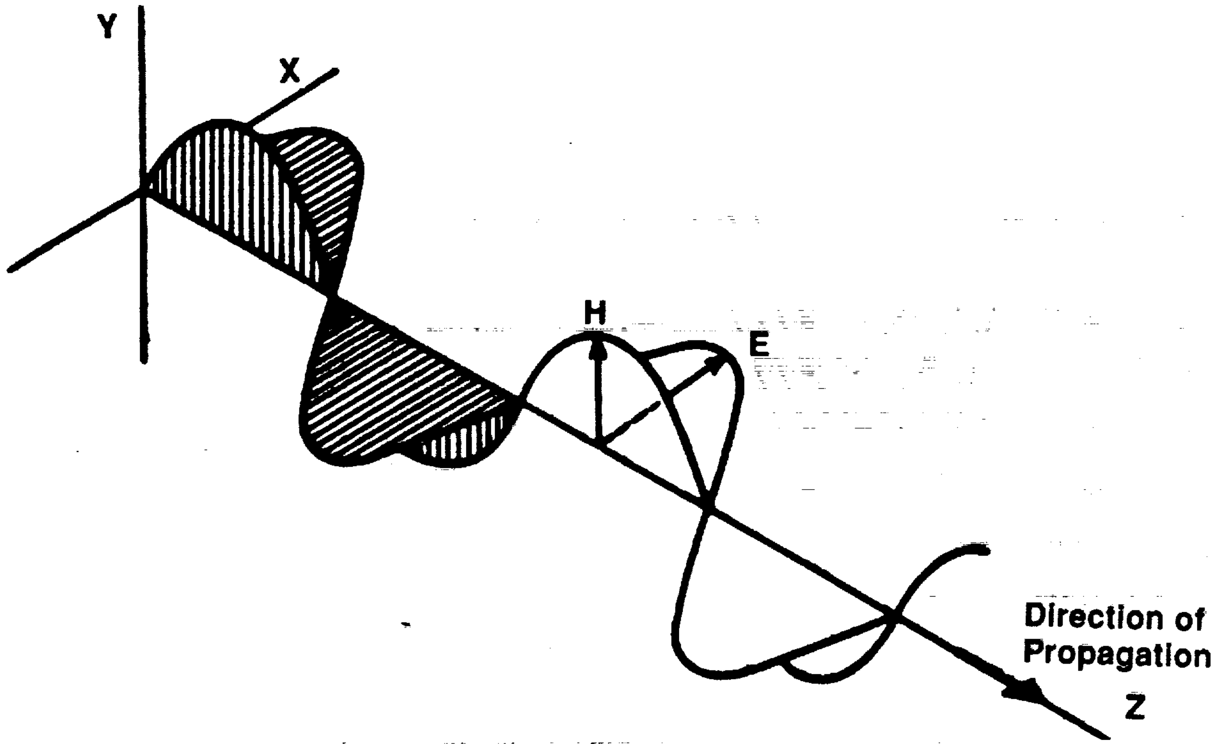


Figure 20. Phase Relationship of Electromagnetic Wave.

Here, α is the attenuation factor and is found as:

$$\alpha = \omega\sqrt{\frac{\mu\varepsilon}{2}\left(\sqrt{1 + \left(\frac{\sigma}{\omega\varepsilon}\right)^2} - 1\right)}$$

and β is the phase shift constant, written as:

$$\beta = \omega\sqrt{\frac{\mu\varepsilon}{2}\left(\sqrt{1 + \left(\frac{\sigma}{\omega\varepsilon}\right)^2} + 1\right)}$$

In the above equations, ω is the frequency of the electromagnetic wave in radians/second, μ is the permeability of the material in henries/inch, ε is the permittivity of the material in farads/inch, and σ is the conductivity of the material in siemens/inch.

The index of refraction of a material is the ratio of the wavelength or phase velocity in free space to that in the material. Since the wavelength can be written in terms of the phase shift constant as:

$$\lambda = \frac{2\pi}{\beta}$$

the index of refraction can be written in terms of the phase shift constant as:

$$n = \frac{\beta}{\beta_o}$$

where β is the phase shift factor for the dielectric material and β_o is the value of the phase shift constant for free space found as

$$\beta_o = \delta\sqrt{\mu_o \varepsilon_o}$$

The calculation of the reflected and transmitted electromagnetic energies is done using Snell's law of refraction and the Fresnel formulas. Snell's law can be stated as:

$$n_1 \sin\theta = n_2 \sin\phi$$

where n_1 and n_2 are the index of refraction of material 1 and 2, θ is the angle of incidence of an electromagnetic wave propagating from material 1 to material 2, and ϕ is the angle of refraction of the transmitted portion of the electromagnetic wave. Figure 21 shows the geometry associated with an impinging electromagnetic wave.

The Fresnel formulas assume an impinging wave with components of the electric vector that are in and out of the plane of incidence defined in Figure 21. This impinging wave is written as:

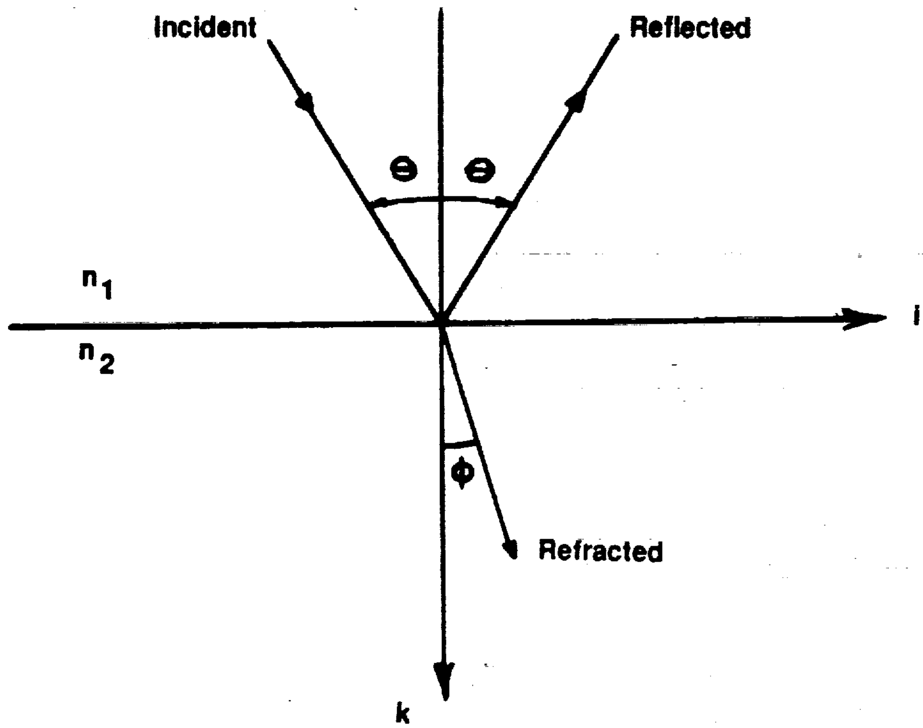


Figure 21. Geometry of Snell's Law.

$$A_1 = A_{p1}(\cos\theta\, I - \sin\theta\, K) + A_{s1} J$$

where A_{p1} is the in plane amplitude and A_{s1} is the out of plane amplitude, and θ is the angle of incidence on the interface between materials 1 and 2. The reflected portion of this wave can then be written:

$$A_2 = A_{p2}(\cos\theta\, I + \sin\theta\, K) + A_{s2} J$$

where

$$A_{p2} = -A_{p1} \frac{\tan(\theta - \phi)}{\tan(\theta + \phi)}$$

$$A_{s2} = -A_{s1} \frac{\sin(\theta - \phi)}{\sin(\theta + \phi)}$$

The refracted wave can be similarly written as:

$$A_3 = A_{p3}(\cos\phi\, I - \sin\phi\, K) + A_{s3} J$$

where

$$A_{p3} = A_{p1} \frac{2 \sin\phi \cos\theta}{\sin(\theta + \phi)\cos(\theta - \phi)}$$

where

$$A_{s3} = A_{s1} \frac{2 \sin\phi \cos\theta}{\sin(\theta + \phi)}$$

For both the reflected and refracted waves, θ is the angle of incidence and ϕ is the angle of refraction obtained using Snell's law.

For normal incidence these equations can be written much more simply using the index of refraction, n. For the reflected wave:

$$A_2 = -A_1 \frac{n - 1}{n + 1}$$

and for the refracted wave:

$$A_3 = A_1 \frac{2}{n + 1}$$

The attenuation (or absorption) of a transmitted wave can be found using the term, $e^{-\alpha x}$, where x is the distance traveled in the material with an attenuation factor of α.

The above calculations for the electromagnetic response were coded into the CSTEM program. Another related feature that was developed is a method to determine what element faces of the model are exposed to an electromagnetic wave. This was done by determining how much of the edges of an element face can be seen from a viewpoint located on the -Z wave coordinate system axis looking in the direction of propagation. Once the exposed faces have been determined, an electromagnetic analysis can be performed for a wave impinging on each exposed face.

The exposure analysis is done by considering the faces of an element where the tests are determined by the corner nodes only. The global coordinates of the corner nodes are rotated to a local system so that the local Z axis lies parallel to the direction of propagation of the incoming electromagnetic waves. The face is split into two triangles and an area coordinate system is set up for each of the triangles. The exposed amount of each side of these triangles is determined by comparing with all other faces. The exposed amount of the face is assumed to be the same as the amount of the boundary of the face which is visible from the viewpoint of the incoming waves.

The frontal area exposed to the waves can be determined by summing over the faces. The frontal area of each face is found as the area projection of the face on the local X-Y coordinate system plane (which is perpendicular to the wave propagation direction). This area is then modified according to the amount of the face visible to the incoming waves, and summed into the total exposed frontal area.

The computer program WAVES has been successfully installed into the CSTEM analysis system. This program calculates the reflection and transmission of electromagnetic waves given a stackup sequence of materials and their electromagnetic properties. Similar to the other electromagnetic analyzers in CSTEM, WAVES will calculate a single cross section at a time. Multiple cross sections are handled as separate calculations as was done previously. The material properties needed are the real and complex parts of the relative permittivity and permeability.

A description of the electromagnetic analysis portion of the CSTEM code as is currently implemented is included as Table 2. This description is a summary in words of the steps and the order in which they are followed in performing an electromagnetic analysis with the wave equations. The alternate table lookup procedure is not a part of this description. Included in

Table 2. Description of Absorption Code.

ROUTINE	
ABMAN	
ABINP2	Read waves and paths to be calculated (input sheets)
ABMAN	Read # of absorption load cases to be done.
ABMAN	LOOP over load cases.
ABMAN	: Read wave-path combination for this load case
TRANIO	: Get transformation defining wave coord system
ABMAN	: LOOP over paths
ABMAN	: : Get starting element and face for this path
(EXPOSE)	(this is specified or from exposure analysis)
ABMAN	: : Get parameters describing incident wave (magnitude, freq, polar. angle, free space prop's)
ELEMIO	: : Get element information
ABMAN	: : LOOP over elements in path
ELGEOM	: : : Get element geometry
ELDISP	: : : Update nodal coords with displacements
ABSCAL	: : : Calculate properties entering element
ABSCAL	: : : Check element face parallel to layers
ABSCAL	: : : Initialize thickness coordinate
ABSCAL	: : : Set up unit vectors for propagation and E field directions, including polarization.
ABSCAL	: : : LOOP over layers.
ABSCAL	: : : : Get layer mat'l properties, thickness
ABSCAL	: : : : Determine elem coord system axis thru thickness.
ABSCAL	: : : : (Integration loop here, but not really doing an integration. Calculations done at centroid.)
S20IF	: : : : Form shape functions (at centroid).
JACOB	: : : : Compute Jacobian (relates elem system to global)
ABSCAL	: : : : Calculate global coordinates at layer upper and lower surface (at centroid).
ABSCAL	: : : : Calculate thickness of layer.
LAYROT	: : : : Calculate global orientation of mat'l properties
ABSCAL	: : : : Check for same mat'l and orientation. If so, electric properties are same so attenuate only.
CROSS	: : : : Calculate an inward normal to impinged surface.
DOT	: : : : Find angle between surface and impinging ray.
ABSCAL	: : : : Check for normal incidence (to save time).
CROSS	: : : : Form a normal to plane of incidence. (plane of incidence is defined by propagation direction and surface normal)
CROSS	: : : : Set up an incidence coord system.
DOT	: : : : Find angle between E and plane of incidence.
ABSCAL	: : : : Decompose E to in plane/out of plane components.
ABSTEM	: : : : Interpolate electric properties for temp., freq.
IMPEDE	: : : : Calculate impedance (ETA).
AB	: : : : Calculate phase shift and attenuation constants
ABSCAL	: : : : (Integration loop ends here)
ABSCAL	: : : : Form index of refraction and angle of refraction
ABSCAL	: : : : Calculate amount of E reflected and refracted.
ABSCAL	: : : : Find distance travelled by refracted E (flat)
A10U8	: : : : Calculate amount refracted E is attenuated.
ABSCAL	: : : : Update thickness, exiting mat'l electric prop's.
ABSCAL	: : : LOOP END (layers)
ABSCAL	: : : Update outgoing E magnitude, orientation.
ABMAN	: : : Check for error (bad face or bad impinge angle)
LNEXT	: : : Find elements adjacent to present element.
ELPATH	: : : Find element connected to present element exit face
ABMAN	: : LOOP END (elements)
ABMAN	: : Print out results for this path.
ABMAN	: LOOP END (paths)
ABMAN	LOOP END (load cases)

the description are the subroutines where each step is performed. It is also easy to see the number of times a given step must be done in this type of analysis.

2.3 CSTEM Acoustic Capability

The capability was incorporated into CSTEM to determine acoustic characteristics due to structural vibration. The approach being to determine the radiation efficiencies of the structure for each vibration mode as a function of frequency. The structure geometry and mode shapes are obtained from an eignevalue run of a finite element model. Once the radiation efficiencies have been found, the overall acoustic characteristics are determined by a summation of the contribution of the sound levels generated by each mode, given a prescribed force on the structure. Two methods for computing eigenvalues and eigenvectors have been implemented in CSTEM, the secant method determinant search and subspace iteration. These capabilities have been validated and verified against close form solutions.

The CSTEM calculation of the sound power produced by a vibrating structure begins with the determination of the surface of the structure. The surface of the structure is found automatically from the element connectivities. In addition, a portion of the surface can be eliminated from a sound power calculations by masking it out. This masking is useful if only a portion of the surface is considered to be radiating sound.

The eigenvalues and natural frequencies of the structure are first found. The radiation efficiencies for each mode are then found as a function of frequency based on the mode shapes and geometry of the structure. Once the radiation efficiencies are known, the sound power produced by a forced vibration of the structure can then be found by a modal summation of the contribution of each mode.

The sound power calculated is based on the equation developed by Lord Rayleigh. This integral equation may be applied to extended plane surfaces. Other analysts have used this equation for nonplanar structures with success (Reference 1) for determining the radiation efficiencies and sound power. Radiation efficiency and sound power calculations used in CSTEM are based on References 2 and 3.

The radiation efficiency calculations have been verified with a test case of rectangular plate. The results of running the model in CSTEM and that produced by Wallace are shown in Figure 22.

2.4 CSTEM Tailoring

The CSTEM tailoring capability is built on the STAEBL (Structural Tailoring Of Engine Blades) computer program from NASA Lewis. This was developed to perform engine fan and compressor blade numerical optimizations.

The program consists of two major modules; CONMIN, which performs the optimization and ANALIZ, which supplies values for the parameter involved in the optimization.

Sound Power Calculation Input Description.

LINE	NUMOMG	NDRMSK	NODDIR	NODFRC

NUMOMG — Number of frequencies that radiation efficiencies and sound power are calculated

NDRMSK — Number of mask points

NODDIR — Direction of nodal forcing function
X direction = 1
Y direction = 2
Z direction = 3

NODFRC — Node number at which force is applied

LINE	FROMIN	FROMAX

FROMIN — Minimum frequency for sound power calculation (Hz)

FROMAX — Maximum frequency for sound power calculation (Hz)

The NUMONG frequencies will be logarithmically distributed between FROMIN and FROMAX.

LINE	FORCE

FORCE — The magnitude of the applied force (lbf)

LINE	ETA(1)	ETA(2)	ETA(NEIG)

ETA(i) — The modal loss factor for each mode. One value required for each eigenvalue calculated.

LINE	DIRMSK(1)	DIRMSK(2)	DIRMSK(3)

DIRMSK(i) — The X, Y, and Z coordinates of the masking point.

A masking point is used to eliminate some of the surfaces of elements from being included in the sound power calculations. Faces of elements interior to a structure are automatically eliminated by the program and need not be considered for masking.

It is often desirable to not include all surfaces of a structure for sound calculations. For example the exterior surface of a pipe might be the only surface desired for sound power calculations. The interior of the pipe can be masked out by specifying a masking point anywhere in the interior of the pipe.

Masked surfaces are defined to be any element surface which has a positive dot product between the surface of an element and a vector from the element face to the masked point. Thus only one point is required to mask all interior surface faces of a straight pipe.

Output of radiation efficiencies are dimensionless and the sound power is output in WATTS.

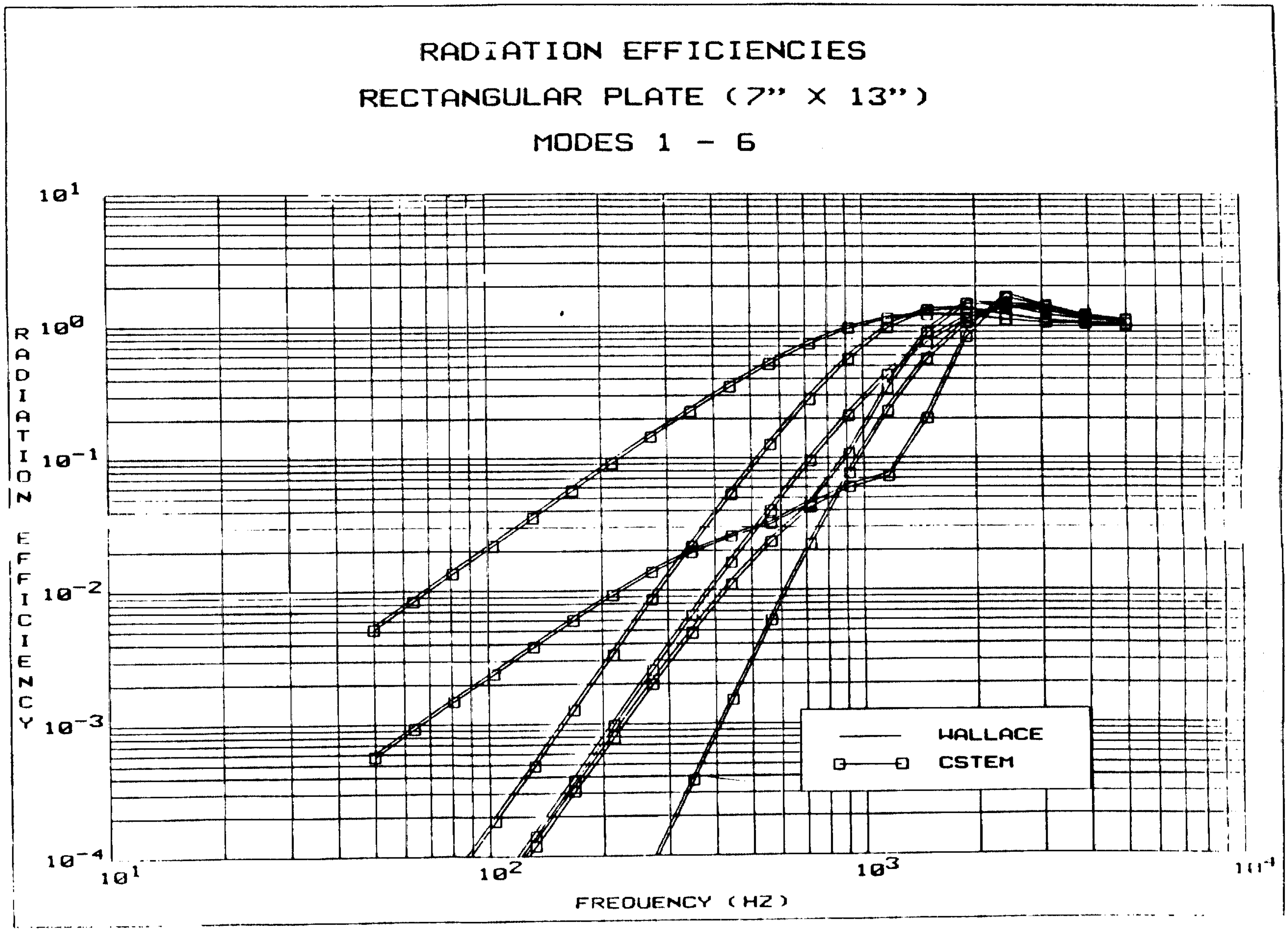


Figure 22. Correlation Between CSTEM Predictions and Closed Form Solution for the Radiation Efficiencies of a Rectangular Plate.

CONMIN works on a set of up to 125 global variables to perform the optimization. A subset of these are selected by input and designated as design parameters, constraints, or object functions. These variables are quantities such as blade thickness, layer orientation, root chord length, and blade weight.

A main driver program, COPES, calls CONMIN initially to set up the global variables. It then calls ANALIZ which reads input and computes a sufficient set of parameters for optimization to precede. CONMIN supplies ANALIZ with a modified set of input and the procedure continues until some convergence criteria is met.

The CONMIN module was abstracted from STAEBL and coupled with CSTEM structural, thermal, electromagnetic, and acoustic application modules. This was successfully debugged and the following verification/validation cases have been run.

The first case involved the tailoring of the natural frequencies of a plate. In this, seven global variables were used. Three of these were the first three natural frequencies and the other four were ply angles of the material that made up the model. The model used was a cantilevered plate consisting of nine 20-noded elements with a total of 96 nodes. The material was a 4 ply composite material.

In the optimization input, the lowest frequency was taken as the objective function to be minimized by varying the 4 ply angles and constrained by the other 2 frequencies. After 10 iterations, the object function was reduced from 565 CPS to 461 CPS. Each iteration took about 10 seconds on a Cray XMP.

The CSTEM tailoring program is capable of tailoring a finite element model to control the noise produced by the structure.

A 96-noded 9-element square plate model was used as a test case. The noise was in response to 4000 DB 565 cps source. The plate was constructed of 4 plies of composite material at different orientation angles. These angles were taken as design variables. Their variation leads to the change to natural frequencies and associated mode shapes of the plate.

As shown in Table 3 the noise level (the objective function) was reduced by a factor of 33 after 24 iterations (200 seconds on the CRAY XMP).

Table 3. Sound Power Tailoring.

Iteration Number	Ply Angles				Frequencies, cps			Noise, watts
1	45.0	90	90	45.0	5650	1447	3074	7.1×10^{-6}
2	67/5	90	90	45	674	1403	2892	1.8×10^{-6}
5	45	90	90	67.5	674	1403	2892	1.8×10^{-6}
10	67.5	45	90	67.5	734	1435	2782	4.8×10^{-7}
15	39	90	90	79	734	1256	2721	2.2×10^{-6}
20	79	90	90	39.6	734	1256	2721	2.2×10^{-6}
23	90	90	90	80	1007	1377	2437	2.16×10^{-7}
24	79	90	90	79	981	1393	2473	2.13×10^{-7}

An example tailoring problem was run with the CSTEM electromagnetic analyzer using commercially available electrical properties for composite materials (see Table 4). These properties are for a single frequency and temperature only and are given in a different form than needed by the electromagnetic analyzer. It is assumed that these properties are typical of those that the CSTEM analyzer may be used for, so the program was modified to accept electrical material data given as dielectric constant and loss tangent. The permittivity, permeability, and conductivity needed by the electromagnetic analyzer can be obtained from these properties.

Table 4. Electrical Properties Comparison.

Material Fiber/Resin	Loss Dielectric Constant	Tangent, 10^{-4}
Spectra/Polyester	2.20	7
Spectra/Epoxy	2.12	20
Spectra/Vinylester	2.16	21
Spectra/Polyethylene	1.99	17
Glass/Polyester	4.30	150
Aramid/Polyester	3.50	500

The permeability when not mentioned is assumed to be that of free space (1.257E-6 henry/meter or 3.192E-8 henry/inch). The permittivity can be found using

$$\varepsilon = \varepsilon_r \varepsilon_o$$

where ε_r is the given relative dielectric constant and ε_o is the permittivity of free space (8.854E-12 farad/meter or 2.249E-13 farad/inch). The conductivity is found from the equation for the loss tangent as

$$\tan \delta = \frac{\sigma}{\omega\varepsilon}$$

where ω is the frequency and σ is the conductivity.

The amount of electromagnetic energy transmitted through a 1/2 inch thickness of material was calculated for a series of impingement angles. The results are shown in Figure 23. It can be seen that the greatest amount of transmitted energy is at normal incidence. The difference in material types is also evident.

Table 5 gives the derived electromagnetic properties ε and $\sigma(\mu = 1.0)$ for these materials. The range in properties, particularly since σ is a function of frequency, will make for significant optimization problems.

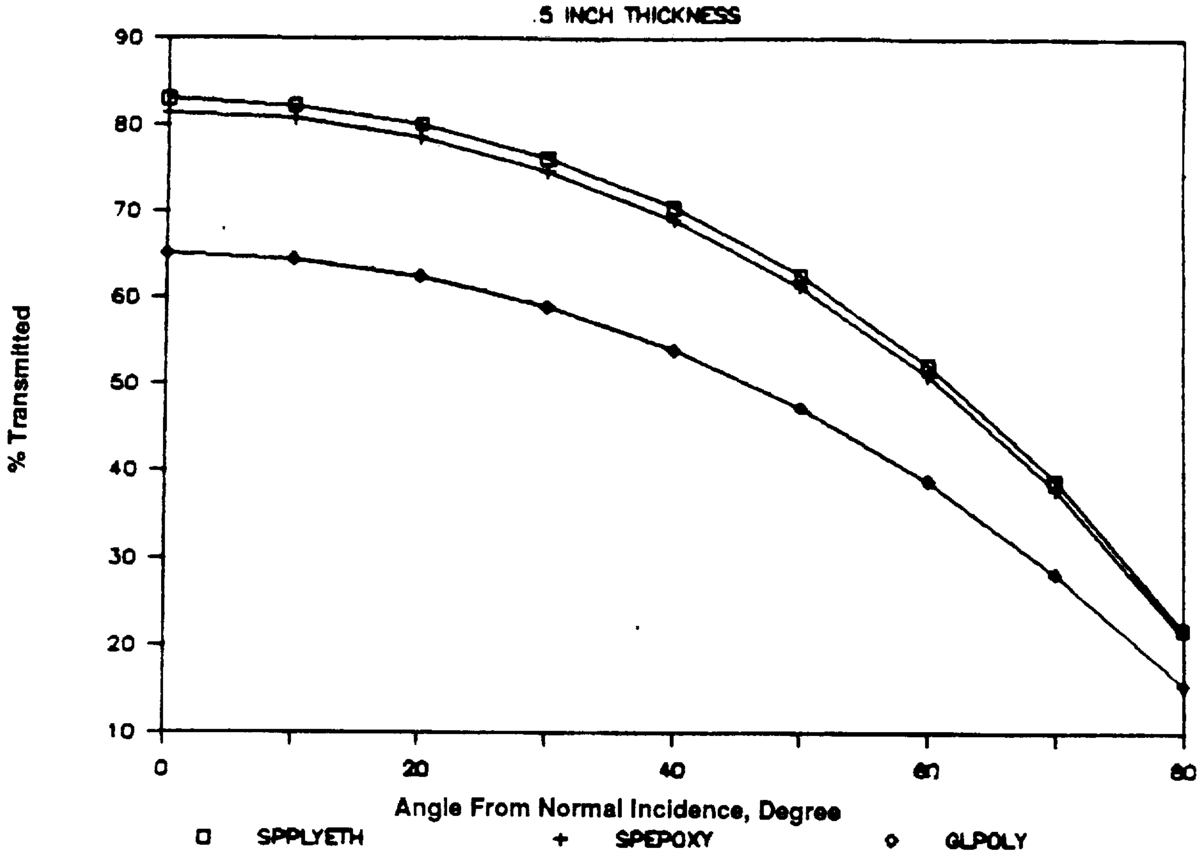


Figure 23. Amount of Electromagnetic Energy Transmitted.

Table 5. Derived Electromagnetic Properties.

Material Fiber/Resin	Permittivity $\varepsilon(10^{13})$ farad/inch	Conductivity at 10 GHz $\sigma(10^{-7})$ Siemens/inch
Spectra/Polyester	4.95	34.36
Spectra/Epoxy	4.77	95.36
Spectra/Vinylester	4.86	102.01
Spectra/Polyethylene	4.48	76.08
Glass/Polyester	9.67	1450.61
Aramid/Polyester	7.87	3935.75

As a test case for electromagnetic absorption, a cantilever beam of eight 8-noded bricks was used. The beam was constructed of two layers of different materials (orthogonal to the X-Y plane). These materials were a SPECTRA fiber in an epoxy matrix (SPEPOXY) and ARAMID fibers in a polyester matrix (ARAPOLY). Two design variables (the percentage of

thickness of each material) were used. The absorption in the Z direction was the objective function to be maximized. The results are as shown in Table 6.

Table 6. Electromagnetic Absorption Tailoring.

SPEPOXY, %	50	54	45	25	33
ARAPOLY, %	50	46	55	75	77
Absorption, %	9.3	8.63	10.14	13.48	13.82

In an effort to determine how to improve the efficiency of the CSTEM optimizer, discussions were held with Dr. J.K. Casey, an applied mathematician at GE who has experience working in optimization problems. One concern was that a great deal of computational effort would be required in order to optimize all of the different aspects of a CSTEM problem.

Dr. Casey pointed out that much of the efficiency in an optimization problem in the hands of the user of the program. To efficiently optimize a problem the user should decompose it and optimize a single variable separately in several one dimensional problems rather than try to optimize several variables in one multiple dimension problem. The order in which the optimization is carried out can be chosen to maximize the performance gained when optimizing a variable at a time. The user should choose to first optimize those variables that experience shows to be most critical.

Also, a simple, coarse mesh should be used when optimizing. If a problem is sensitive to mesh density it is possible to use a coarse model to get derivative values that the optimizer needs and a finer model to compute the results.

Presently, the optimizer in CSTEM can only work on a single object function. This is the same as the STAEBL program that the optimization package in CSTEM comes from. A way to optimize multiple object functions is being examined. This would require an additional executive program around the optimization loop.

One possibility would be to direct the optimizer to perform an iteration on each of the object functions and store results from the separate optimizing iterations. An additional decision module would also be required to determine the averaged direction to take which would make some improvement in all of the object functions. This would continue until arriving at a point which requires opposite changes in order to improve the different object functions.

Another way to look at multiple object functions might be to optimize each separately, using previously optimized object functions as constraints. This would require prioritizing the object functions, iterating on the most important one first then using it as a constraining value while iterating on a second object function. It should be pointed out that our meeting with Dr. Casey indicated that it may be better to decompose the problem and optimize each objective as a separate problem.

To prepare for more complicated optimization work a large 8-noded brick model of a J-85 compressor blade was recactivated (Figures 24 and 25). This model consists of 100 8-noded isoparametric elements and 242 nodes. The bill-of-material for this blade is titanium. To benchmark the model and further verify the large deformation logic in CSTEM, comparison runs were made among the centrifugal stiffening predictions of NASTRAN (MSC), MASS (GEAE), and the updated Lagrangian approach in CSTEM. A rotor speed of 16,000 rpm was assumed and titanium material properties. The lowest three natural frequencies were computed by each program as shown in Table 7.

Table 7. Correlation of CSTEM Nonlinear Eigenanalysis.

Mode	CSTEM Frequency, cps	MASS Frequency	NASTRAN Frequency, cps
1	936	935	981
2	2763	2763	2092
3	3265	3278	3302

CSTEM correlates excellently with MASS which has been related with test data NASTRAN gives consistently stiffer predictions.

The use of CSTEM in the analysis of these fairly large blade models has pointed out the need for an alternative method to generate element layers in a model. The present manual layer input is an element by element specification which requires the user to manually determine where element boundaries lie within a layup. This can be difficult.

Another method has now been included in CSTEM which allows the user to specify a layup and then apply it to a cross section of elements. To specify the cross section the user need only input the surface element and the element axis that points through the thickness (along the cross section). The program will determine what elements are in the cross section and the thickness of the elements at their centroids. Using the element thicknesses and the thicknesses of the layers comprising the layup the program determines how many layers lie within each element. These element layers may then be copied to other similar cross sections by specifying the beginning element of each of these cross sections.

A UDF blade model has been acquired for CSTEM development. This propulsor blade model consisted of 756 nodes and 480 eight-noded isoparametric elements. The model was run previously on our in-house program MASS using orthotropic material properties. This model was run on the CRAY version of CSTEM. The composite properties were simulated by 23 different materials using 481 reference nodes to define the composite directions. The large displacement analysis capability of CSTEM was used to account for centrifugal stiffening. Six load increments were used to spool up from flight idle to cruise. At the cruise condition, a deformed position eigenanalysis was performed using the subspace iteration technique. The lowest three natural frequencies were obtained and are compared in Table 8 against MASS.

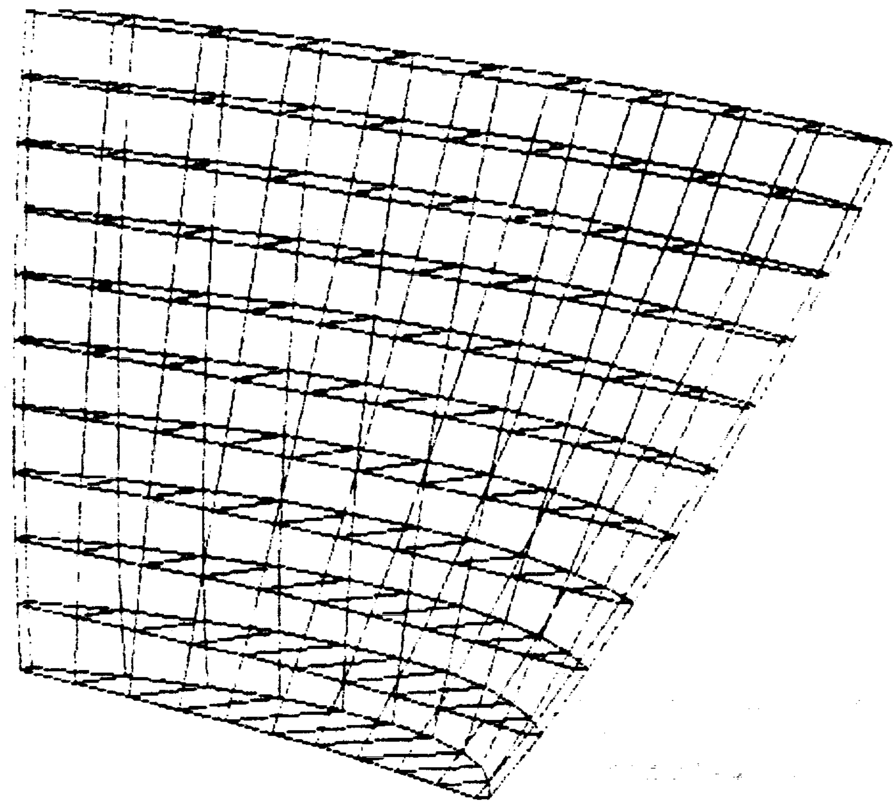


Figure 24. View 1 of J85 Compressor Blade Finite Element Model.

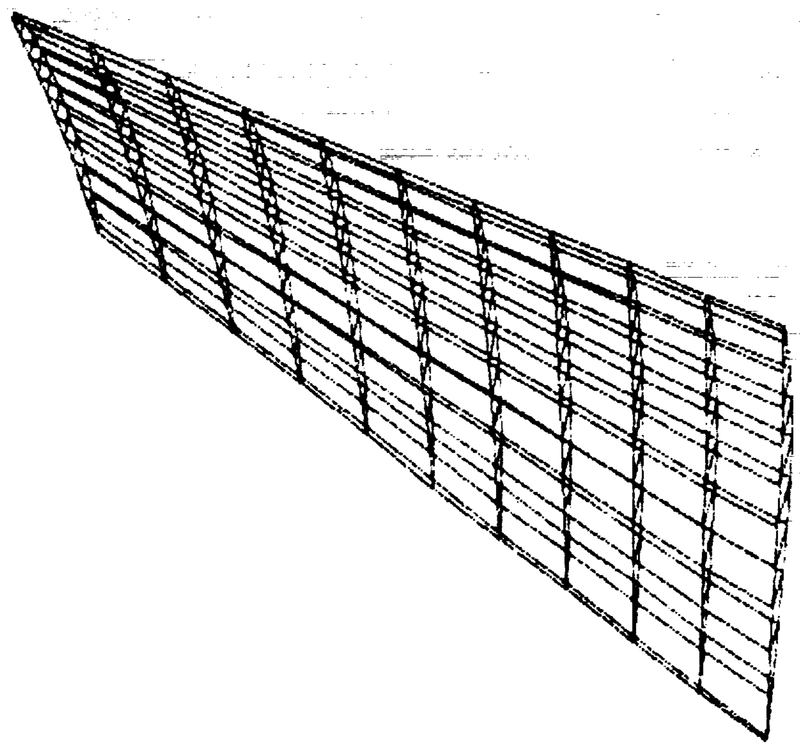


Figure 25. View 2 of J85 Compressor Blade Finite Element Model.

Table 8. Deformed Position Eigenanalysis With Subspace Iteration.

	CSTEM	Mass
1.	281	278
2.	593	598
3.	858	853

This required approximate 370 CPU seconds on the CRAY.

2.5 CSTEM Composite Micromechanics Using ICAN

Composite micromechanics analyses in the CSTEM program are performed using a modified version of the ICAN computer program obtained from NASA-Lewis. This program and the inputs are described in the ICAN User's Manual. In the CSTEM application these inputs are obtained from the CSTEM input deck, are calculated from the results of the CSTEM finite element program, or are defaulted to a value such that a certain type of analysis is always performed. The following is an attempt to describe the ICAN installation in the CSTEM program and how the specific ICAN inputs are obtained or calculated.

The ICAN routines can be accessed in two separate ways. This results from the fact that the ICAN program itself incorporates routines from a program called INHYD which calculates layer properties from the constituent fiber and matrix properties. These ICAN/INHYD type routines can be accessed to generate elastic material properties for composite systems whose constituent properties are contained in a data bank. Section X.4 in the CSTEM input sheets describe the input variables for this type of use. The actual micromechanics analysis can be performed as well using the ICAN routines. In this usage, the elastic material properties must be obtained through the use of the ICAN/INHYD routines contained in ICAN as mentioned above. Since ICAN performs its analysis for conditions at a cross section of the laminate, additional input specifying this location is needed. Section XVIII. in the CSTEM input sheets describe the inputs specifying these cross sections.

The inputs needed to use ICAN to generate elastic properties are those contained in the Ply Details cards and Material System Details cards described in the ICAN User's Manual. The material identification number is assigned consecutively in order of input. All properties are returned in the local material coordinate system and are incorporated into the global system by the CSTEM program so that the orientation angle and thickness in the Ply Details card group are not necessary. The use temperature is passed from the CSTEM program to the ICAN/INHYD routines and is interpolated from nodal temperature values. There is a check for a trash temperature value of 1234567. in which case a reference temperature of 70°F is used. At present this would occur only if using the linear heat transfer analysis option without a corresponding structural analysis.

The ICAN materials data bank is assumed to exist as a file called 'ICANBNK' on the machine where the CSTEM program is being run. This file is opened by the CSTEM program

(subroutine RDCAN) when the option for ICAN generation of material properties is activated.

The use of ICAN as part of the CSTEM program for micromechanics analysis requires only that the material properties be generated by ICAN as previously described, that the location(s) where the analysis applies be specified, and whether strains and curvatures or loads are to be calculated from the finite element results as input for the ICAN analyzer. The ICAN analysis is done immediately after the structural results have been computed and printed out so that an ICAN load case corresponds to a structural load case. However, an ICAN analysis can be done for up to 10 different cross sections in a load case.

The cross sections within the structure where the ICAN analysis is to be performed are specified by listing the element numbers through the cross section starting from the bottom. The calculations to obtain the necessary information for ICAN are done at the centroids of the layer surfaces within the listed elements. Therefore, the cross section element centroids should be lined up normal to the layers. A maximum of 10 elements are allowed in defining a cross section.

The number of layers in the cross section are determined by counting the number of layers in each consecutive element of the cross section starting from the bottom of the laminate (the first element listed). There is no check for layers extending between elements, so if a layer lies in two adjacent elements it would be treated as two layers having the same orientation. The present limit on number of layers through the cross section is 100.

The limits on number of cross sections, elements through the cross section, and layers through the cross section are all contained in PARAMETER statements. Therefore, they can be fairly easily changed by modifying this PARAMETER statement and recompiling the CSTEM program.

The analysis options set by the booleans described in the ICAN User's Manual are hardwired in the program (Table 9).

Table 9. ICAN Boolean Settings.

COMSAT	True
BIDE	True
CSANB	False
NONUDF	True

The boolean RINDV is determined by the CSTEM input variable ICAN. RINDV is True if ICAN = 1 and False if ICAN = 2.

Much of the data required by ICAN as described in the Ply Details cards and Material System Details cards of the ICAN User's Manual is already available from the generation of the material properties using the ICAN/INHYD routines. The percent of moisture of each

layer, which was input for the generation of material properties, is transferred to storage in array PL, row 72 for use by ICAN.

The use temperature for each layer is interpolated from the temperature of the element nodes to the midsurface of the layer using the element shape functions. ICAN uses the difference in this use temperature and the cure temperature of the material. This value is stored for each ply in array PL, row 50.

The thickness of each layer is calculated by first locating the centroid of the upper and lower surfaces of the layer using the element shape functions and nodal coordinates then calculating the distance between these two points. This value is stored for each layer in array PL, row 7.

The orientation angle of the layer is obtained from the element layer information stored using the LAYIO routine of the CSTEM program. This angle is placed in the THLC vector used by the ICAN routines.

The bulk of the calculations done using the finite element results are to obtain the loadings for the ICAN analyzer. These calculations assume that the strain varies linearly through the cross section, while the stress may be discontinuous. The loadings can be either strains and curvatures or loads in the form of stress resultants and couples. Strains and stresses at the cross section element integration points, along with the element geometry information are the quantities used from the finite element analysis. The integration points are generally in a Gaussian distribution on each layer midsurface. In the case of an element with only one layer the integration points will be in a three dimensional Gaussian distribution.

For the strain and curvature loading option of ICAN, the strains and curvatures at the cross section midsurface (reference plane) are needed. The strains are calculated by first calculating the thickness through the entire cross section as the distance between the centroid on the upper surface of the cross section and the centroid on the lower surface. These centroids are calculated using shape functions and nodal coordinates. Once the cross section thickness has been determined the layer thicknesses are summed beginning from the bottom surface until this sum exceeds half of the cross section thickness. The strains at the integration points of the element in which the reference plane lies are then interpolated to the reference plane centroid using Lagrange interpolation functions.

The reference plane curvature is calculated using the equation for strain from laminated plate theory which can be written as:

$$\{\varepsilon\} = \{\varepsilon^\circ\} - Z\,\{K^\circ\}$$

where $\{\varepsilon^\circ\}$ is the reference plane strain, $\{K^\circ\}$ is the reference plane curvature, and Z is the distance from the reference plane to the location where the strain, $\{\varepsilon\}$, is desired. This equation can be solved for the curvatures.

$$\{K^\circ\} = 1/Z\,(\{\varepsilon^\circ\} - \{\varepsilon\})$$

The curvatures can be calculated using this equation, the strain at the centroid of the top layer of the cross section, and the distance from the reference plane to this centroid.

The in-plane stress resultants or membrane loads used by ICAN can be calculated using the equations

$$N_x = \int_{-h/2}^{h/2} \sigma_x \, dz$$

$$N_y = \int_{-h/2}^{h/2} \sigma_y \, dz$$

$$N_{xy} = \int_{-h/2}^{h/2} \sigma_{xy} \, dz$$

while the equations for the stress couples or bending resultants are

$$M_x = \int_{-h/2}^{h/2} \sigma_x z \, dz$$

$$M_y = \int_{-h/2}^{h/2} \sigma_y \, z \, dz$$

$$M_{xy} = \int_{-h/2}^{h/2} \sigma_{xy} \, z \, dz$$

and the equations for the transverse shear resultants are

$$Q_x = \int_{-h/2}^{h/2} \sigma_{xz}\, dz$$

$$Q_y = \int_{h/2}^{h/2} \sigma_{yz}\, dz$$

In these equations, h is the cross section thickness and Z is the distance from the reference plane.

Since the stress through the cross section is assumed to be piecewise linear these equations must be integrated piecewise, integrating over each individual layer and summing the layer results over the cross section. For the membrane loads this can be written in general as

$$\{N\} = \sum_{\text{layers}} \int_{z_r - \frac{h_l}{2}}^{z_r + \frac{h_l}{2}} \{\sigma\}\, dz$$

where Z_r is the distance from the reference plane to the layer midplane and h is the layer thickness. Assuming the stress to be linear over the layer it can be written

$$\{\sigma\} = \{\sigma_{mid}\} + \frac{ze}{hl} (\{\sigma_{top}\} - \{\sigma_{bot}\})$$

where $\{\sigma_{mid}\}$ is the layer midsurface stress, $\{\sigma_{top}\}$ is the stress at the layer top surface, $\{\sigma_{bot}\}$ is the stress at the layer bottom surface, and Z is the distance from the layer midsurface to a point in the layer and can be written

$$Zl = Z - Z_r$$

Substituting this stress function into the integral gives

$$\{N\} = \sum_{\text{layers}} \left[\{\sigma_{mid}\} \int_{z_r - \frac{he}{2}}^{z_r + \frac{he}{2}} dz + \frac{\{\sigma_{top} - \sigma_{bot}\}}{h_e} zdz - \frac{\{\sigma_{top} - \sigma_{bot}\}}{h_e} Z_r \int_{z_r - \frac{h_e}{2}}^{z_r + \frac{hl}{2}} dz \right]$$

which when integrated results in the following general equation for the membrane loads.

$$\{N\} = \sum_{\text{layers}} \{\sigma_{mid}\}\, h_l$$

The transverse shear resultant integrals are of the same form as those for the membrane loads. The bending resultants are somewhat different and use of the same procedure results in a general equation for the bending resultants.

$$\{M\} = \sum_{\text{layers}} \left[\{\sigma_{mid}\}\, Z_r\, h_l + \frac{\{\sigma_{top} - \sigma_{bot}\}}{12}\, h_l^z \right]$$

The quantities needed in the equations for the loads are obtained by looping through all the layers beginning from the bottom of the cross section, calculating the contribution of that layer to the loads and summing. The stresses at the middle, top, and bottom surface centroids are calculated from the strains at these same locations. The strains are interpolated from all the integration point locations within the element using Lagrange interpolation functions. Temperatures at the middle, top, and bottom surface centroids are interpolated from the element nodes using the element shape functions. The material matrices are then calculated so that the stress at these points can then be obtained from the strains.

2.6 Multiple Layer Elements in CSTEM

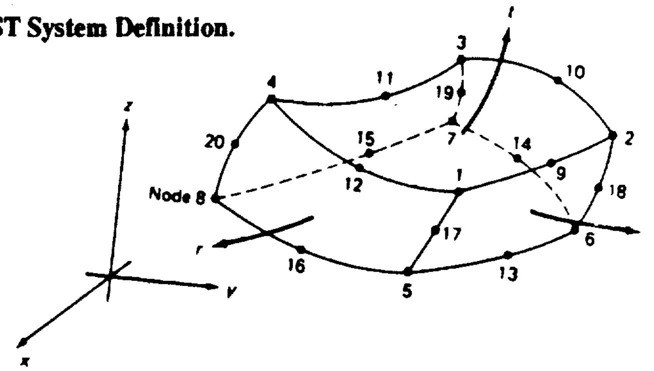
The information necessary to define the layering of an element is the thickness axis, the layer material, the layer thickness, the number of layers, and the orientation of the layer.

Multiple layers/materials in CSTEM elements require that careful attention is paid to the element local coordinate system. This element local coordinate system (also called the rst system) defines the isoparametric space of the element and is determined by the element connectivity. In its isoparametric space an element is a 2r by 2s by 2t cube with the rst system origin at the centroid. In other words the element sides (or faces) are planes perpendicular to one of the rst axes and parallel to the plane made up by the other two axes. These element faces lie at the +1 and -1 locations of its perpendicular rst axis. The rst system definition for CSTEM is shown in Table 10.

Material layers within an element are set up so that the upper and lower surfaces of a layer are parallel to a pair of element faces in isoparametric space. This means that, in isoparametric space, the layer surfaces are parallel to a plane defined by two rst axes with the third axis being perpendicular to the layer surfaces, The perpendicular rst axis defines the thickness direction

Table 10. RST System Definition.

Axis	Corner Nodes			
+t	1	2	3	4
-t	5	6	7	8
+s	1	2	6	5
-s	3	4	8	7
+r	1	4	8	5
-r	2	3	7	6



so that going through the element thickness means we are going perpendicular to the layers. This way of defining element layers means that when modeling a layered structure the mesh must follow the layer surfaces so that layers run between opposite element faces. Layers can not cut diagonally across an element.

The layer material is defined by a material number and the properties associated with this material are stored by reference to this material number. Layer materials are treated as orthotropic materials, so isotropic materials can be used since they are a subset of orthotropics. Orthotropic materials have directionality so a material coordinate system is associated with each material. The material properties are specified with respect to this material coordinate system with the material X or 1 axis usually being the principle or stronger direction, although this is not a requirement.

Since the layers are of constant thickness in isoparametric space, the layer thicknesses are stored as fractions of the element thickness. Therefore, if the element varies in thickness then so do the layers. CSTEM is set up so that layer number 1 is that the negative end of the thickness axis. The layers are then numbered consecutively progressing along the thickness axis in the positive direction.

There are two ways to define the orientation of the layer with respect to the global coordinate system. One is to define a transformation directly from the material coordinate system to the global coordinate system. In CSTEM a particular transformation has a number associated with it much like the materials. For global orientation of element layers the transformation number is all that is required and the transformation associated with that number is used to rotate the material properties directly to global.

Many structures are of complex shapes and curvatures which makes it difficult to specify the material orientation with respect to a global coordinate system. Since the elements in a mesh generally follow the shape of a structure (as do the layers) it may be much easier to specify the material orientation with respect to the element coordinate system. The material orientation with respect to the global system can then be calculated since the information relating the rst system to the global system is available from the element geometry definition. Using this method in CSTEM assumes that the material coordinate system is aligned so that the material Z or 3 axis is coincident with the element thickness axis. For local orientation of element layers the angle of rotation about the material 3 axis is all that is required. The

reference axis for this rotation angle depends on which of the rst axes is defined as the thickness axis. Table 11 shows the reference (0 degree) orientation of the material coordinate system to the element rst system for each thickness axis case.

Table 11. Unrotated Material Orientation with Element.

	Material Axis	Element Axes		
	1	r	s	t
	2	s	t	r
(thickness axis)	3	t	r	s

The orientation of the element coordinate system with respect to the global coordinate system is contained in the Jacobian matrix. The Jacobian contains both the angular relation of the element coordinate system to the global coordinate system as well as the stretching or shrinking of the element axes in relation to the global axes. For the material orientation only the angular relationship is required. Therefore, the Jacobian must be normalized row by row to get the transformation matrix from the element coordinate system to the global coordinate system. (If the Jacobian were to be normalized column by column the transformation matrix from the global coordinate system to the element coordinate system would be obtained). A possible drawback in using the Jacobian in this way is that the rst coordinate system may not be an orthogonal system when viewed in the global coordinate system. This would occur when element corners are not right angles.

The rotation of the material matrix from the material coordinate system to the global coordinate system is calculated as:

$$[D]^g = [T]^T \, [D]^1 \, [T]$$

Therefore, to get the complete transformation matrix from the material coordinate system to the global coordinate system the normalized Jacobian (which rotates from element to global) is premultiplied by the planar transformation matrix which rotates the material coordinate system to the element coordinate system.

Further example problems using composite materials are being run with the structural portion of the CSTEM code to gain experience with the layered element. In many composite structures the number of plies through the thickness varies along the structure. Test cases are being prepared to determine the best way to model this ply dropoff using the CSTEM layered element, which must contain the same number of layers throughout the element.

Figure 26 shows an example of a cross section geometry which changes from four layers through the thickness to two layers through the thickness. Two methods can be used to model this with multiple layer elements. In the first method, the elements are tapered to follow the geometry of the structure. This results in a mismatch of layers between one element to another

(Figure 27). This poses no computational difficulties since the stiffness for each element is generated separately. In the second method the element geometry would not follow the actual structure geometry, but would be modeled as if there was no drop off of plies. The ply drop off is modeled by zeroing out the material properties of the absent layers (Figure 28). Again this does not cause a computational problem since the layers that are present will add their stiffness to the element. A combination of these two methods might be the most effective in a general problem.

A tapered composite cantilever beam example was used initially to examine this problem. A model with only one layer per element was used to validate the results of modeling this beam with multiple layer elements (see Figures 29 and 30).

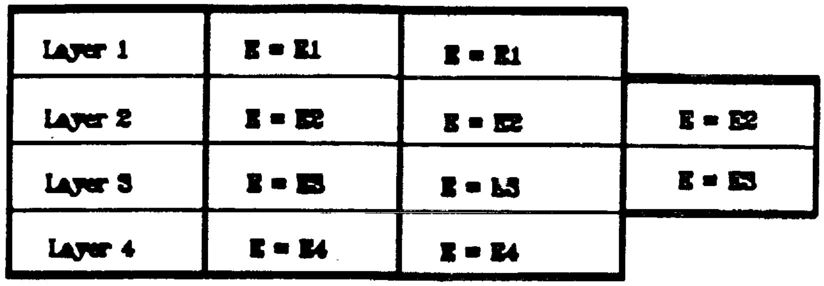


Figure 26. Actual Cross Section Geometry.

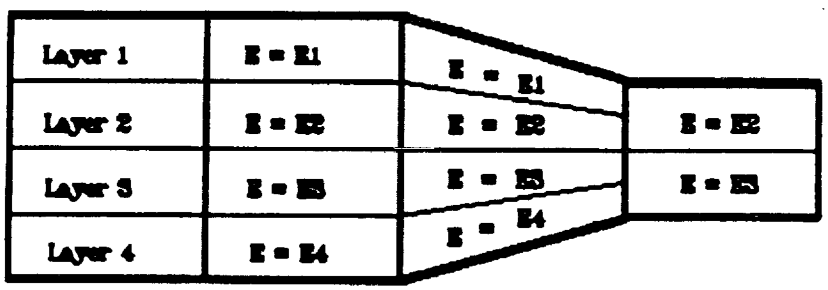


Figure 27. Tapered Elements.

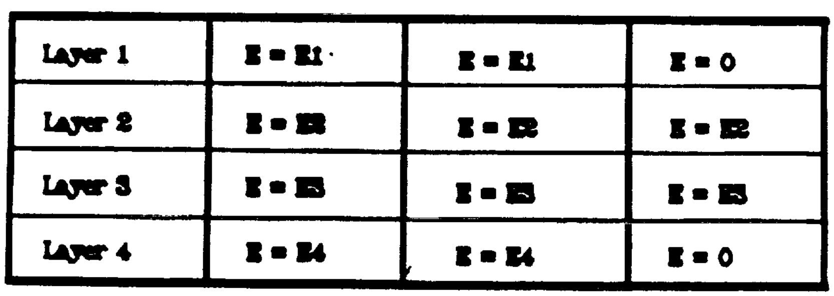


Figure 28. Zero Material.

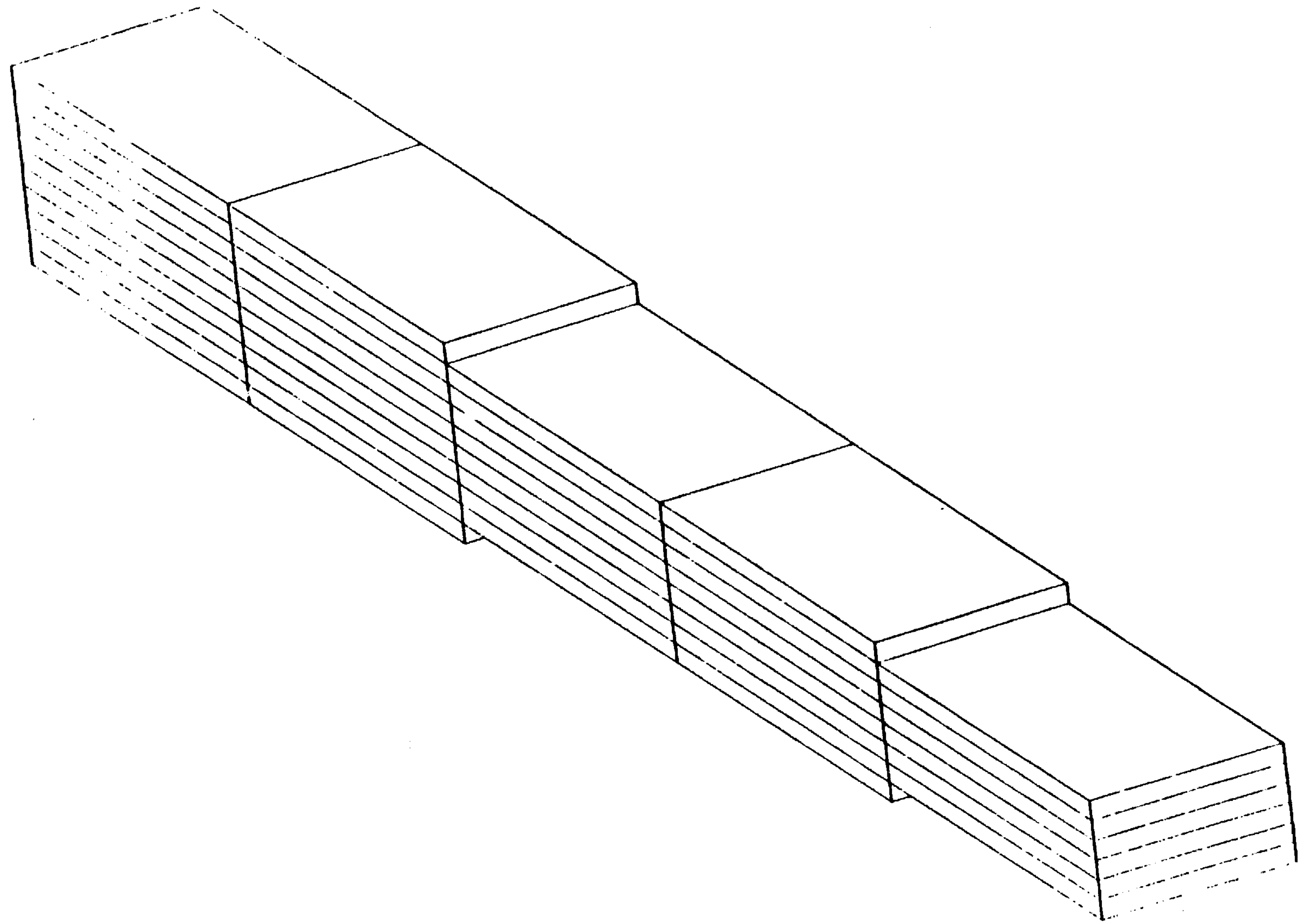


Figure 29. One Element Per Layer.

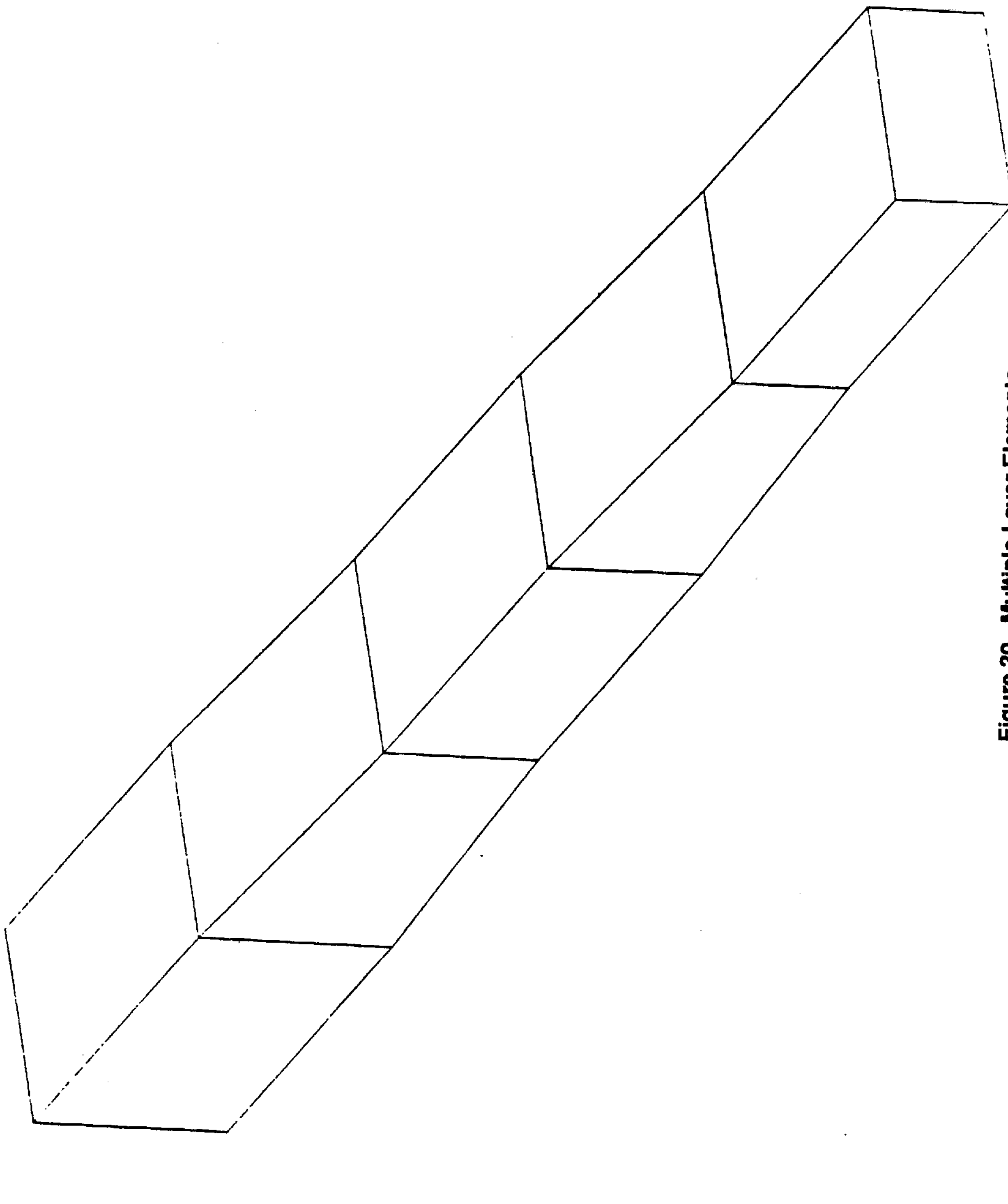


Figure 30. Multiple Layer Elements.

3.0 Conclusions

The first five tasks of the NASA Statement of Work has been successfully completed. Unique graded material finite elements have been developed with structural, thermal, electromagnetic, and acoustic capabilities. A CSTEM analyzer has been developed which performs coupled structural/thermal/electromagnetic/acoustic analysis. Other advanced capabilities are the large deformation analysis of graded composite structures, deformed position eigenanalysis, transient nonlinear heat transfer with right-hand-side pseudo-fluxes, electromagnetic absorption, and sound power predictions. A CSTEM tailoring capability has been developed by marrying the optimization routines from the NASA supplied STAEBL program with the CSTEM application routines. Verification and validation cases have been run. One final task remains, the CSTEM tailoring of simulated components.

References

1. Hayes, P.E., "Determining Vibration, Radiation Efficiency, and Noise Characteristics of Structural Designs Using Analytical Techniques," Proceedings - Society of Automotive Engineers P-106, SAE Warrendale, PA, pp 263-270, 1982).

2. Wallace, C.E., "Radiation Resistance of a Rectangular Panel," The Journal of the Acoustical Society of America, Vol. 51, No. 3 (Part 2), pp 946-952, 1972.

3. Fahy, Frank, "Sound and Structural Vibration Radiation, Transmission and Response," Academic Press, London, 1985.

REPORT DOCUMENTATION PAGE

Form Approved
OMB No. 0704-0188

1. AGENCY USE ONLY (*Leave blank*)	2. REPORT DATE	3. REPORT TYPE AND DATES COVERED
	April 1992	Final Contractor Report

4. TITLE AND SUBTITLE

Coupled Structural/Thermal/Electromagnetic Analysis/Tailoring of Graded Composite Structures
Third Annual Status Report

5. FUNDING NUMBERS

WU-505-62-91
C-NAS3-24538

6. AUTHOR(S)

R.L. McKnight, H. Huang, M. Hartle, R. Holt, and D. Carpenter

7. PERFORMING ORGANIZATION NAME(S) AND ADDRESS(ES)

General Electric Aircraft Engines
1 Neumann Way
Cincinnati, Ohio 45215

8. PERFORMING ORGANIZATION REPORT NUMBER

None

9. SPONSORING/MONITORING AGENCY NAMES(S) AND ADDRESS(ES)

National Aeronautics and Space Administration
Lewis Research Center
Cleveland, Ohio 44135–3191

10. SPONSORING/MONITORING AGENCY REPORT NUMBER

NASA CR–189152

11. SUPPLEMENTARY NOTES

Project Manager, C. Chamis, Structures Division, NASA Lewis Research Center, (216) 433–3252.

12a. DISTRIBUTION/AVAILABILITY STATEMENT

Unclassified - Unlimited
Subject Category 39

12b. DISTRIBUTION CODE

13. ABSTRACT ***(Maximum 200 words)***

Accomplishments are described for the fourth years effort of a 5-year program to develop a methodology for coupled structural/thermal/electromagnetic analysis/tailoring of graded component structures. These accomplishments include: (1) demonstration of coupled solution capability; (2) alternate CSTEM electromagnetic technology; (3) CSTEM acoustic capability; (4) CSTEM tailoring; (5) CSTEM composite micromechanics using ICAN; (6) Multiple layer elements in CSTEM.

14. SUBJECT TERMS

FEM structural analysis; FEM thermal analysis; FEM electromagnetic analysis; Graded composites; Coupled analysis; Optimization; Tailoring

15. NUMBER OF PAGES

56

16. PRICE CODE

A04

17. SECURITY CLASSIFICATION OF REPORT	18. SECURITY CLASSIFICATION OF THIS PAGE	19. SECURITY CLASSIFICATION OF ABSTRACT	20. LIMITATION OF ABSTRACT
Unclassified	Unclassified	Unclassified	

NSN 7540-01-280-5500

U.S. GOVERNMENT PRINTING OFFICE: 1992 - 750-034/60306

Standard Form 298 (Rev. 2-89)
Prescribed by ANSI Std. Z39-18
298-102